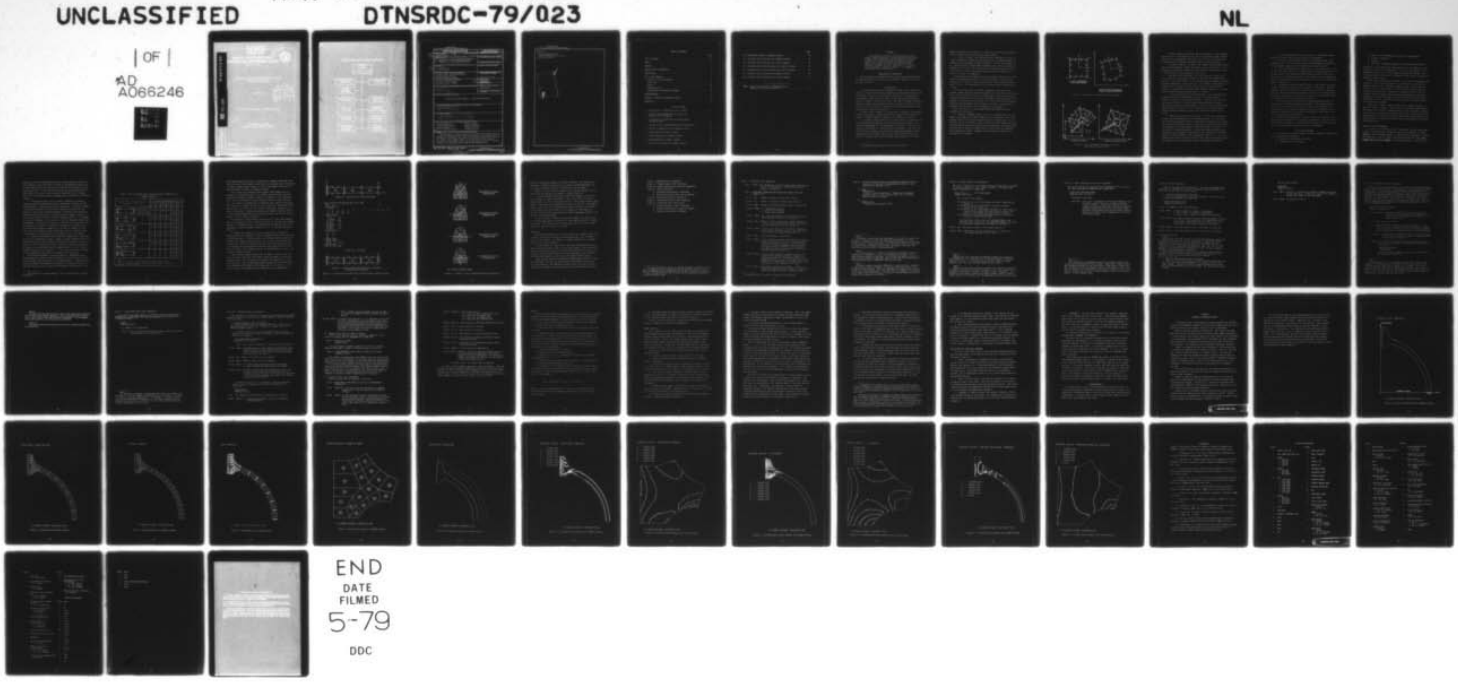
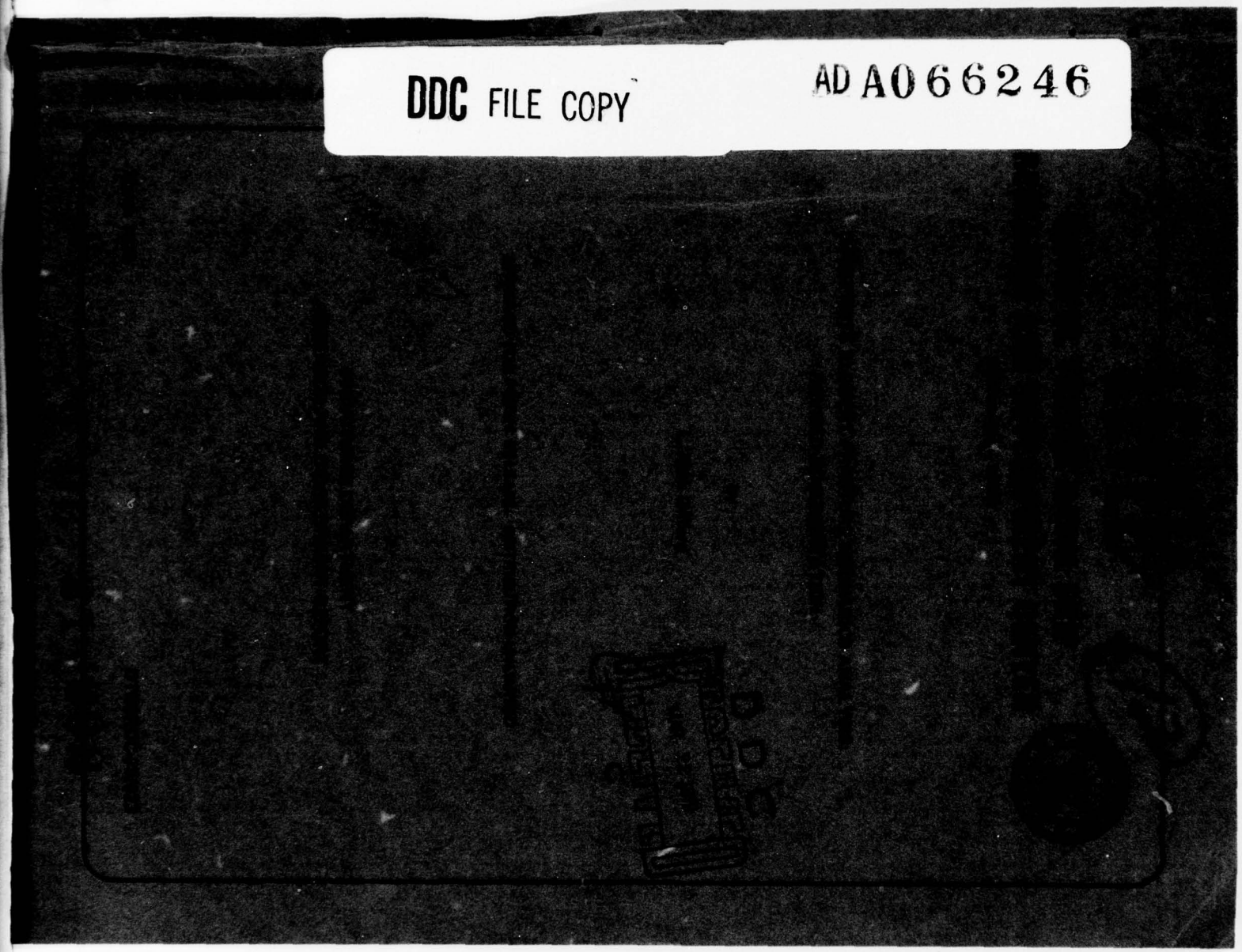
AD-A066 246 DAVID W TAYLOR NAVAL SHIP RESEARCH AND DEVELOPMENT CE--ETC F/G 20/11
APES-FINITE ELEMENT FRACTURE MECHANICS ANALYSIS: REVISED DOCUME--ETC(U)
MAR 79 L N GIFFORD

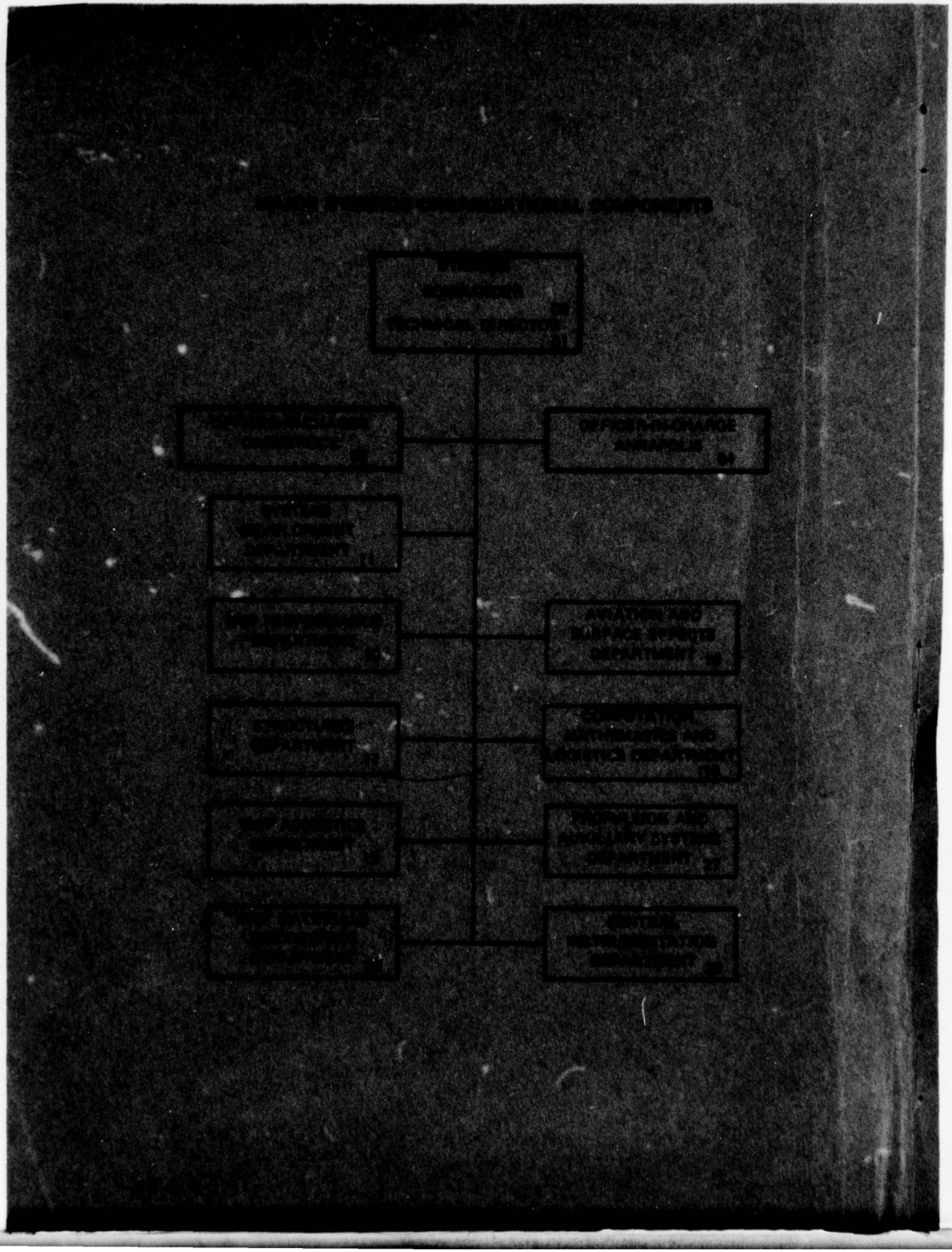
UNCLASSIFIED DTNSRDC-79/023 NL

| OF |
AD
A066246



END
DATE
FILMED
5-79
DDC





UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE

READ INSTRUCTIONS BEFORE COMPLETING FORM

(14) 1. REPORT NUMBER: DTNSRDC-79/023 | 2. GOVT ACCESSION NO. (9) Research and development rept. | 3. RECIPIENT'S CATALOG NUMBER

4. TITLE (and Subtitle): (6) APES-FINITE ELEMENT FRACTURE MECHANICS ANALYSIS: REVISED DOCUMENTATION | 5. TYPE OF REPORT & PERIOD COVERED

6. PERFORMING ORG. REPORT NUMBER

(10) 7. AUTHOR(s): L. Nash Gifford | 8. CONTRACT OR GRANT NUMBER(s)

9. PERFORMING ORGANIZATION NAME AND ADDRESS
David W. Taylor Naval Ship Research and Development Center
Bethesda, Maryland 20084

10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS
(See reverse side)

11. CONTROLLING OFFICE NAME AND ADDRESS
Naval Sea Systems Command
Washington, D.C. 20362

(11) 12. REPORT DATE: March 1979

13. NUMBER OF PAGES: 56

14. MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office)
(12) 55p.

15. SECURITY CLASS. (of this report): UNCLASSIFIED

15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

APPROVED FOR PUBLIC RELEASE: DISTRIBUTION UNLIMITED

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)
(16) F43422

18. SUPPLEMENTARY NOTES
(17) SF43422592

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

Fracture Mechanics	Stress Analysis
Finite-Element Methods	Numerical Methods
Singular Elements	Cracked Structures
Stress Intensity Factors	Fatigue Analysis

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

New information is given concerning the APES finite element computer program for fracture mechanics and stress analysis of two-dimensional and axisymmetric structures. Described are useful new capabilities which have been added to the program as well as the additional input parameters necessary for their application. The report takes the form of a complete revision of the appendix of the original APES documentation.

DD FORM 1 JAN 73 1473 EDITION OF 1 NOV 65 IS OBSOLETE
S/N 0102-LF-014-6601

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

384682

(Block 10)

Program Element 62543N
NAVSEA SUBPROJECT SF 43.422.592
Work Unit 1720-592

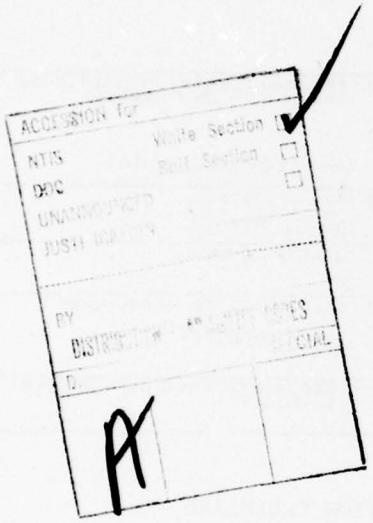


TABLE OF CONTENTS

LIST OF FIGURES

ABSTRACT

New information is given concerning the APES finite element computer program for fracture mechanics and stress analysis of two-dimensional and axisymmetric structures. Described are useful new capabilities which have been added to the program as well as the additional input parameters necessary for their application. The report takes the form of a complete revision of the appendix of the original APES documentation.

ADMINISTRATIVE INFORMATION

The developments described herein were authorized and funded within the Submarine Structures Exploratory Development Program SF 43.422.592, Work Unit 1720-592.

INTRODUCTION

The acronym APES stands for Axisymmetric/Planar Elastic Structures, which is a finite element computer program, developed specifically to provide linear, elastic, fracture mechanics analysis for two-dimensional (axisymmetric, plane strain, or plane stress) structures containing a crack or a number of interacting cracks. The mode I (opening) and mode II (inplane sliding) stress-intensity factors K_I and K_{II} are directly calculated at the crack tip(s) in addition to the usual quantities of nodal displacements, strains, and stresses.

The basic finite element is the high-order, (bicubic) 12-node quadrilateral isoparametric element described by Zienkiewicz[1*] and shown in Figure 1. Fracture mechanics capability is included in two ways.

1. Enriched 12-node elements are available which, in addition to the usual displacement assumption, also include the singular modes of deformation appropriate for the displacement field near a crack tip. The crack tip corresponds to a corner node on such elements. The stress intensity factors K_I and K_{II} are carried as additional unknowns directly calculated by the program. This technique, described in detail elsewhere,[2,3] eliminates the need for extreme mesh refinement or any other

*A complete listing of references is given on page 47.

special treatment in the vicinity of crack tips; K_I and K_{II} are predicted quite accurately even with relatively coarse meshes.

2. A small circular "core" element[4] may be centered at the tip of a single crack and be joined along its periphery to an arbitrary number of standard 12-node elements as shown in Figure 2. In the core element, K_I, K_{II}, and the crack tip displacement components are carried as unknowns. Although this treatment also yields accurate, directly calculated values for K_I and K_{II}, the enriched-element approach described previously is easier to use and is recommended.

Although APES was designed for fracture mechanics problems, the high order and accompanying accuracy of the basic element is such that the program is also frequently used in non-fracture, stress analysis applications. There is no requirement that the structure under consideration be cracked.

It is extremely easy to use APES, compared to most other finite element computer programs, primarily because there are relatively few input data. The high order of the basic element makes it possible to solve an "average" problem by using from 20 to 40 elements. Thus, when APES can be applied, it has become a very popular tool. This popularity has prompted additions and improvements to the program in the past 3 years which have culminated in the present need to document the new features.

Since publication of the original APES program,[2] the following features have been added.

1. Graphical (CALCOMP) Output. Plots of the idealization, with and without node numbers and element numbers, can be obtained by using a preprocessor which also performs logical checks on the input data. In the analysis portion of the APES system, an overlay has been added which plots the idealization, deflected structure, and contour plots (across the whole structure and in selected "zoomup" areas) of selected stress components and/or temperature values. Examples of this graphical capability are provided in the appendix.

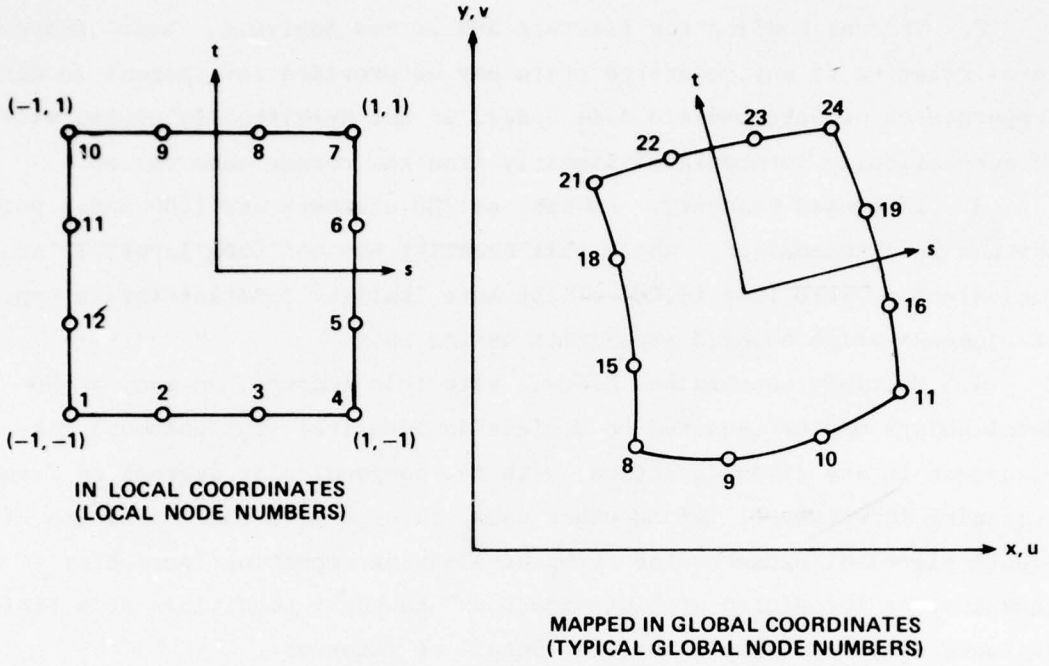


Figure 1 - Quadrilateral 12-Node Basic Finite Element

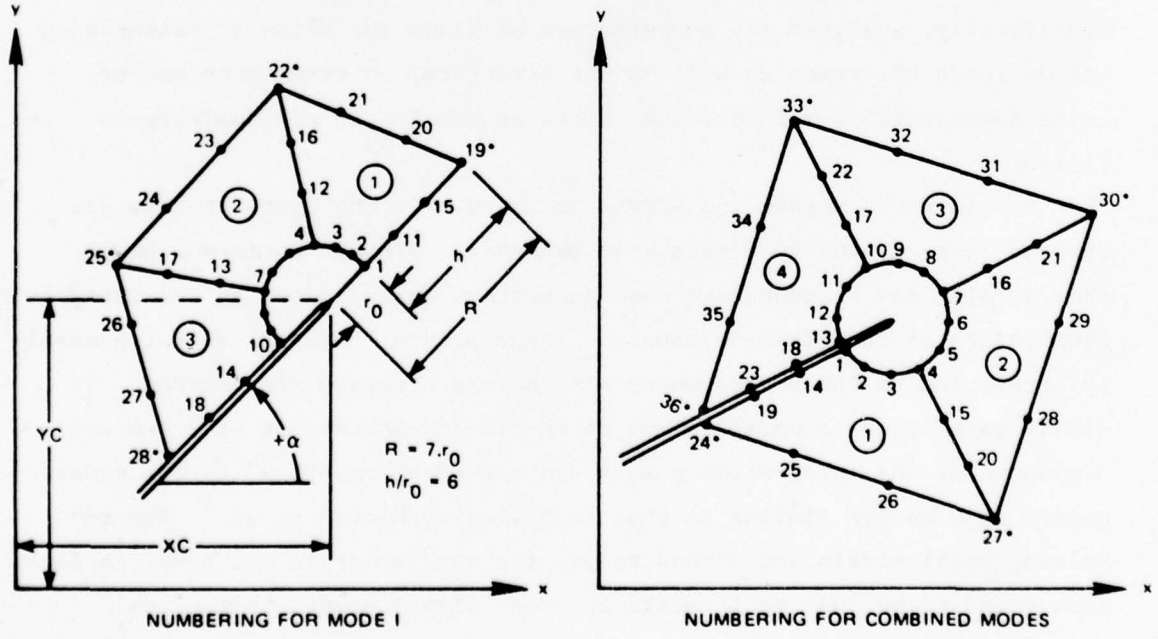


Figure 2 - Node and Element Numbering for Special Core Element Crack Tip Models

2. Thermal Loading for Fracture and Stress Analysis. Nodal temperatures relative to any reference state may be provided for thermal loading. Temperatures of intermediate side nodes, if not specifically given, will be automatically interpolated linearly from the corner node values.

3. Increased Capacity. As many as 200 elements and 1400 nodal points may now be accommodated. While this quantity may not seem large, it can be equivalent to more than 10,000 of the more familiar constant-stress types of elements which enjoyed popularity in the past.

4. Multiply Constrained Nodes. With this feature, as many as 30 nodal points may be required to achieve an identical (but unknown) displacement in any given direction, with the perpendicular degrees of freedom remaining independent. Among other uses, this permits the imposition of double planes of symmetry for structures having repeating (periodic) geometry, or imposition of "infinite body" boundary conditions at a finite distance away from some structural detail of interest.

5. Crack Face Loading for Enriched Elements. The consistent nodal loading resulting from distributed tractions on enriched elements is different from that of conventional elements. Incorporation of this feature gives capability not found in many fracture mechanics analyses. Specifically, analyses may be performed of flaws for which a pressurizing medium loads the crack as well as the structure, of crack problems by using superposition methods, and of cracks growing in residual stress fields.

6. Improved Strain and Stress Accuracy. In the past, strains and stresses were evaluated directly at the nodal points. It is now known that strains and stresses are more accurate when evaluated at the integration points of the finite elements. These points, however, do not generally correspond to the points where strains and stresses are desired. To obtain more accurate nodal values of strain and stress, strains are now evaluated at the integration points and are then "smoothed" to the nodal points in a manner similar to that suggested by Hinton et al.[5] The resulting nodal strain and stress values are more accurate and have, in fact, been found to be just as accurate as nodal displacements themselves.[6]

7. Selective Suppression of Certain Printout. Element-by-element printing of strains and/or stresses may now be suppressed.

8. Printout of Nodal Reaction Forces at Constrained Nodes. This information is of use in analyzing the way in which applied loading on a highly redundant structure is distributed to the support points.

9. Further Simplification of Input Data. It is now possible to number only the corner nodes of an idealization and to define elements by corner node number only, leaving the APES program to provide numbers automatically for intermediate side nodes. This feature can save significant effort, and further decreases the probability of error in the already small number of input data.

10. Triangular Elements Having Nine Nodes. Such elements are quite helpful in making a transition from a coarse to a finer mesh. However, no thorough study has yet been made to ascertain the accuracy of these elements (as they are presently formulated) relative to the 12-node element. Preliminary indications are that they are inferior in their present form and, therefore, should be used only with caution.

The previously outlined changes have been implemented without invalidating the original documentation of APES.[2] The user having no interest in these additions can use the old manual. The purpose of the present report is to provide an updated version of the appendix of the original documentation, which discussed preparation of the idealization and the input data for the program.

This update incorporates all additions to APES as of December 1978, and replaces the appendix of the original documentation. This report does not stand alone, however. To understand and use APES effectively, References 2 and 3 should be consulted for theory and examples.

USE OF APES PROGRAM

To use any finite element program, one must provide the computer with the following basic information:

1. Properties of structural material(s)
2. Coordinates of nodal points

3. Manner in which elements are connected to nodal points
4. Boundary conditions
5. Loading
6. Additional information required for specialized applications.

Coordinates of the nodal points and connectivity of the elements to them constitute the bulk of the input data. It is here that manpower costs become high and that most errors (sometimes undetected) are made. Consequently, anything that reduces or simplifies these data will bring substantial economic returns.

Lower costs are particularly possible in this age of third-generation computers, in which literally millions of computations can be made for pennies; computer costs, compared to manpower costs, are very small for most finite element applications. For this reason, a high-order finite element becomes desirable. Although the computational effort (done by computer) may be increased, the input data are significantly decreased, resulting in substantial savings of time and money and a greatly reduced probability of error. Moreover, accuracy is greatly increased with the use of high-order elements.

Input data for APES, relative to most finite element programs, are extremely simple to prepare. Few data need to be provided, and considerable effort has been spent in making these as simple as possible. Nonetheless, the user is expected to be able to follow input instructions and to keep in mind a few rules necessitated by the special purpose nature of the computer program. Guidelines are given in the following text for setting up an idealization.

SETTING UP THE IDEALIZATION

1. Lay out the structure to scale, preferably on linear graph paper to help in defining nodal coordinates, in the first quadrant of a RIGHT-HANDED x- and y-coordinate system. If the problem is axisymmetric, situate the right half of the structural cross section so that the y AXIS CORRESPONDS WITH THE AXIS OF SYMMETRY. Use enlarged details or "blowups" where

necessary so that the idealization will be clearly defined. Neatness is very important; extra time taken to define the idealization clearly will be more than repaid in the long run by saving time later looking for and correcting errors. If the possibility exists that more structure will be added later or that the present structural dimensions will be changed, leave room, since negative nodal coordinates are not permitted in the APES program.

2. Divide the structure into a suitable assemblage of elements, keeping in mind that elements should be concentrated in regions of high stress gradient. Avoid excessive element distortion. As a rule of thumb, corner angles should be not less than 45 degrees nor greater than 135 degrees.* Avoid using triangular elements, if possible, particularly in regions where high accuracy is desired, since the triangular element in the present version of APES has not been thoroughly tested. If a singular core crack tip element is to be used at a crack tip (enriched elements are recommended), a blowup will be necessary to show the idealization in this area; Reference 2 should be consulted for guidance as to element size. If enriched quadrilateral elements are to be used at a number of crack tips, each such element must adjoin ONLY ONE crack tip. Enriched crack tip elements may be rather large; see References 2 and 3, for example.

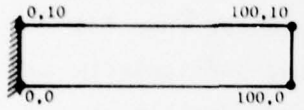
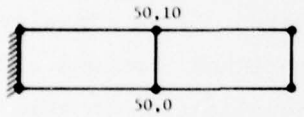
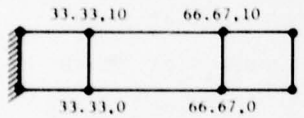
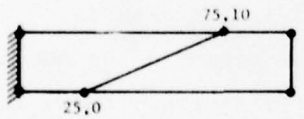
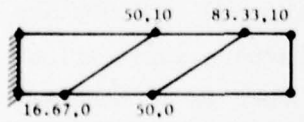
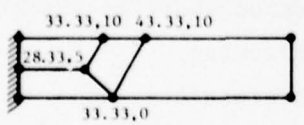
3. Sketch in intermediate nodes along the element edges, using small dots to aid in numbering the nodes. Number the nodes (starting with 1) in ascending numerical order, using any convenient path through the idealization. If a core element with n nodes is used for a fracture application, the nodes on the core element must be numbered from 1 to n in a counterclockwise sense as shown in Figure 2. It is usually unnecessary to number all the intermediate side nodes as described in the following.

APES has been programed to provide numbers for intermediate (noncorner) nodes for elements which are defined (in the input data) by only the node numbers that correspond to the element corners. Although all the nodes may certainly be numbered, if desired, this feature may be used to save time and effort and to reduce the possibility of input errors. The

---

*See Table for an extreme example of loss of accuracy due to element distortion.

TABLE - LOSS OF ACCURACY FOR A CANTILEVER BEAM IN BENDING DUE TO ELEMENT DISTORTION

Element Configuration (Corner Nodes Shown Only) P = 100 lb, E = 1.0 × 10^7 psi, ν = 0.3		Element Type and Integration Order					
		Q-8 (2×2) Ref. 7	Q-8 (3×3) Ref. 8	LST (2-el) Ref. 8	LST (4-el) Ref. 8	Q-12 (3×3) APES	Q-12 (4×4) APES
	N(deg of freedom)	16	16	18	26	24	24
	V(tip deflection)	0.03756	0.03054	0.03001	0.03087	0.03761	0.03761
	σ_T(max tens. stress)					600.0	600.0
	σ_C(max compr. stress)					-600.0	-600.0
	N	26	26	30	46	40	40
	V	0.03872	0.03721	0.03719	0.03804	0.03875	0.03875
	σ_T					600.0	600.0
	σ_C					-600.0	-600.0
	N	36	36	42	66	56	56
	V	0.03922	0.03877	0.03871	0.03935	0.03926	0.03926
	σ_T					600.0	600.0
	σ_C					-600.0	-600.0
	N	26	26	30	46	40	40
	V	0.01463	0.00644	0.03185	0.03202	0.01970	0.01542
	σ_T					668.2	480.2
	σ_C					-394.5	-305.2
	N	36	36	42	66	56	56
	V	0.03112	0.01766	0.03824	0.03845	0.03165	0.02928
	σ_T					859.2	762.7
	σ_C					-602.0	-565.1
	N					66	66
	V					0.03977	0.03941
	σ_T					626.6	613.7
	σ_C					-619.3	-607.2

NOTES:

Beam theory tip deflection: 0.04000 in.; beam theory maximum bending stress: 600.0 psi.

This is an extreme case; if loading is changed to direct uniform compression or tension, results are exact regardless of element distortion.

node number generation feature is automatically triggered when APES recognizes that only 4 node numbers (rather than 12) have been given in the input to define a quadrilateral element, or only 3 node numbers (instead of 9) have been given to define a triangular element.

The user will not generally know in advance what numbers will be assigned to intermediate nodes (of an element defined by only the corner nodes) unless either an adjacent element or elements are fully numbered. As a result, distributed tractions, boundary conditions, edge curvature, etc., for elements containing such intermediate nodes cannot be specified. The solution is to number ALL nodes for an element that contains the intermediate nodes for which tractions, boundary conditions, curvatures, etc., are to be specified. (Elements adjoining fully numbered elements need not be fully numbered themselves and may be specified by only the four corner nodes.) See the example in Figure 3.

Alternatively, an idealization having only the corner nodes numbered may be run through the data checking preprocessor to ascertain the assigned numbers of intermediate nodes. This information may then be used to add appropriate tractions, boundary conditions, etc., to the data deck after this run.

4. Number the elements. Care is necessary since a "frontal" or "mesh annihilation" technique[9] is used to assemble and solve the structural stiffness equations. With this method, the element stiffness matrices are calculated in advancing numerical order as a "front" passes through the element mesh. Degrees of freedom associated with nodes behind the front are eliminated by expressing the stiffness of the structure ahead of the front in terms of the eliminated degrees of freedom behind the front. The process is shown in Figure 4.

Under present program dimensioning, the number of nodes across the front must not exceed 40 for the smaller version of APES, or 80 for the largest version. (If this number is exceeded, the program will so indicate and stop.) Since element numbering determines the size of the front, the elements should be numbered to minimize the size of the front. As a rule of thumb, the elements should be numbered so as to minimize the

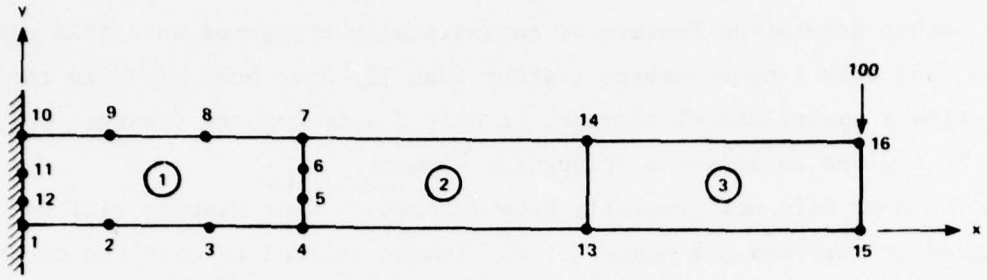


Figure 3a - Idealization for Preparing Input

```
THREE ELEMENT CANTILEVER BEAM TEST CASE
  16     3     1     2                                          3
( BLANK CARD )
    10.E6
    1     1     2     3     4     5     6     7     8     9    10    11    12
    2     4    13    14     7
    3    13    15    16    14
( BLANK CARD )
    1  0.          0.
    4  33.33       0.
    7  33.33       10.
   10  0.          10.
   13  66.67       0.
   14  66.67       10.
   15  100.        0.
   16  100.        10.
( BLANK CARD )
    1     3
   10     3
   11     3
   12     3
( BLANK CARD )
( BLANK CARD )
   16  0.         -100.
( BLANK CARD )
(END OF FILE CARD )
```

Figure 3b - Input Data

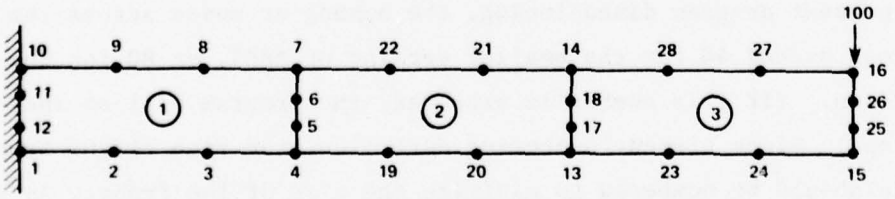
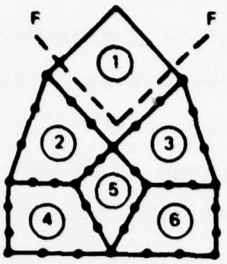
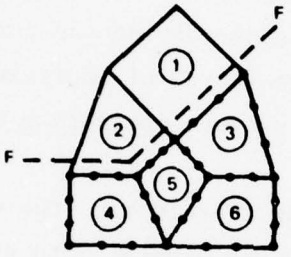


Figure 3c - Computer-Generated Idealization, Including Intermediate Node Numbers

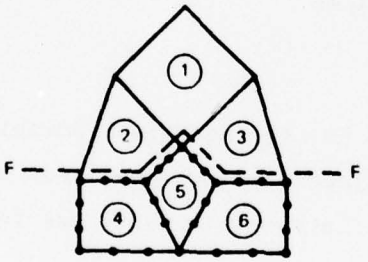
Figure 3 - Idealization and Input Data for Three-Element Cantilever Beam



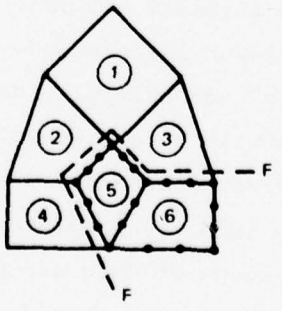
FRONT POSITION AT ELEMENT 1
7 NODES ON FRONT



FRONT POSITION AT ELEMENT 2
10 NODES ON FRONT



FRONT POSITION AT ELEMENT 3
13 NODES ON FRONT



FRONT POSITION AT ELEMENT 4
13 NODES ON FRONT

NOTE: ELIMINATED NODES NOT SHOWN

Figure 4 - Passage of Front through Hypothetical Idealization

difference between the numbers of adjacent elements. This procedure is analogous to numbering nodal points to achieve a minimum bandwidth; see the element numbering of Figure 8 of Reference 2 as an example.

Finally, if one is using a singular core crack tip element, the elements about the core must be numbered as shown in Figure 2.

5. Along straight element edges, APES will automatically generate the coordinates of nodes intermediate to the corner nodes so that the element edges are exactly divided into thirds. Along curved element edges, however, coordinates of intermediate nodes must be provided. This must be done in such a way that the intermediate nodes divide the curved length of the element edge into thirds to a close degree of approximation. Failure to satisfy this requirement will produce inaccurate results in the vicinity of the curved edges.

If one is using a circular core crack tip element, the nodes on the core element must divide the periphery of the element into arcs of identical length. This implies that the angle within the arc of each element adjacent to the core element is the same.

INPUT DATA

Once a good idealization is set up, the rest is a downhill run; it is actually very easy to translate the idealization into a set of numbers to be put in the program. An example of a complete data deck for a simple test case is shown in Figure 3.

At first glance, the input data as described and annotated here may appear extensive; however, they are not really complex. The experienced user has little need to refer to the input instructions which follow. The beginner, however, must "break the ice" and carefully set up a problem or two before the simplicity of the data will become clearly apparent.

Input data are most easily prepared on standard 80-column data sheets. For convenience, the data are divided into seven categorical groups to aid in explanation. The individual groups of data are given as follows in order of appearance in the data deck and are then described in detail.

Group I - Preliminary Data (Required)

Group II - Element Connectivity* (Required)

Group III - Node Coordinate Specification* (Required)

Group IV - Nodal Constraints

- A. Prescribed Displacements* (Required)
- B. Multiply Constrained Node Data (If Needed)

Group V - Distributed Tractions* (Required)

Group VI - Concentrated Nodal Loads* (Required)

Group VII - Additional Data (As Required)

- A. Fracture Mechanics Input (If Needed)
- B. Thermal Stress Analysis Input* (If Needed)
- C. Graphical Output Data (If Needed)

---

*To save the user the effort of counting (possibly erroneously) items to be included in a data group marked with an asterisk, such data groups MUST BE TERMINATED WITH A BLANK CARD; thus Groups V and VI must consist of at least a blank card each, even if there are no distributed tractions or concentrated nodal loads.

Group I - Preliminary Data (Required)

Card 1: TITLE - Any information to be printed with the output such as a title for the problem. (Punch anywhere within Columns 2 through 80.)

Card 2: NN,NE,NMAT,ISTRN,IHEAT,NTIE,INTEG,IDLZ,NFRAC,IPLOT,ISUP
FORMAT (1115)

(1-5) NN - Number of nodes (estimate will suffice).

(6-10) NE - Number of elements (estimate will suffice).

(11-15) NMAT - Number of different materials (max 5, default 1*).

(16-20) ISTRN - 0 = axisymmetric problem.
1 = plane strain problem.
2 = plane stress problem.

(21-25) IHEAT - Not zero means thermal stress analysis is desired. Additional data (Group VII) are required.

(26-30) NTIE - Number of nodes in a multiply constrained node set, if one exists, otherwise leave blank. Additional data are required in Group IV; see Note 1.

(31-35) INTEG - A value of 3 results in 3×3 numerical integration of the element stiffness matrices; any other value will result in 4×4 integration; see Note 2.

(36-40) IDLZ - Presently inactive, leave blank.

(41-45) NFRAC - If not a fracture problem, leave blank. A positive number indicates that a special core element is to be used at a single crack tip. A negative value indicates that enriched elements are to be used in a fracture problem, and NFRAC is the negative of the number of crack tips; see Note 3. Additional data are required in Group VII.

(46-50) IPLOT - A nonzero value indicates graphical output is desired and that additional graphics data will be supplied in Group VII. The value of IPLOT is the maximum dimension in inches of CALCOMP plots. Plots should not be larger than the available plotter bed, allowing for 1.5-inch margins on all sides.

(51-55) ISUP - For optional suppression of output. A value of 1 suppresses element-by-element strains; 2 suppresses element-by-element stresses; 3 suppresses both.

---

*A default value is achieved by leaving the entry blank.

Cards 3: For each different material, two material property cards are required, the first pair corresponding to material 1, the second pair to material 2, etc.

3a: POIS
FORMAT(F10.5)
Poisson's ratio (default 0.3). Because of the default setting, a value of zero cannot be used; use a very small number instead, if required.

3b: E
FORMAT(E10.3)
Young's modulus (default 30.E6).

NOTE 1:

The multiply constrained node option makes it possible to require that a group of nodes have an equal (but unknown) displacement in a given direction. As many as 30 nodes may be included in the multiply constrained (tied) node set. Among other things, this capability permits the imposition of double planes of symmetry which result, for example, when analyzing structures with periodically repeating geometry.

NOTE 2:

For enriched crack tip elements, 8×8 numerical integration (Gaussian quadrature) is automatically used. For conventional elements, 4×4 integration has customarily been used. More recent work suggests that a 3×3 order produces results of comparable accuracy and leads to an approximately 35-percent reduction in computation cost; see the table.

NOTE 3:

To handle cracked structures, NFRAC must be either positive, implying a singular core crack tip element, or negative, indicating the use of enriched crack tip elements. Thus it is impossible to combine both crack tip models in the same idealization. Enriched crack tip elements are recommended; as many as five crack tips may be treated by setting NFRAC equal to the negative of the number of crack tips.

Group II - Element Connectivity (Required)

One card is required for each element, whether conventional or enriched. No card is required for a core crack tip element. This data group MUST BE TERMINATED BY A BLANK CARD.

NEL,N1,N2,N3, ... N11,N12,MAT,THICK
FORMAT (14I5,F10.5)

NEL - Number of the element.

N1 through N12 are the 12 node numbers defining a quadrilateral element, or
N1 through N9 are the 9 node numbers defining a triangular element (N10-N12 blank), or
N1 through N4 are the 4 node numbers defining the corner nodes of a quadrilateral element for which APES is to supply the numbers of intermediate side nodes (N5-N12 blank), or
N1 through N3 are the same as described previously, except for a triangular element (N4-N12 blank).

The node numbers must be listed in COUNTERCLOCKWISE order beginning with a corner node. FOR ENRICHED ELEMENTS, THE FIRST GIVEN NODE MUST CORRESPOND TO THE CRACK TIP; SEE NOTE 4.

(66-70) MAT - The material number of the element (default 1).

(71-80) THICK - Meaningful for piane problems only, the thickness of the element (default 1.0); see Note 5.

NOTE 4:
BECAUSE THE FIRST GIVEN NODE FOR ENRICHED ELEMENTS MUST CORRESPOND TO THE CRACK TIP, it is impossible for an enriched element to join more than one crack tip. If a circular core crack tip element is present (Figure 2) the first node defining Element 1 must be node 4.

NOTE 5:
Unless the planar structure has a stepwise discontinuity in thickness, the default value of unity is most convenient to use. The thickness of a crack tip core element, if present, is taken as that of Element 1. The thickness of each planar element is presently assumed constant; however, a smooth variation in thickness could be implemented, were the need to arise.

Group III - Node Coordinate Specification (Required)

One card is required for each node whose coordinates are to be input. This data group MUST BE TERMINATED BY A BLANK CARD.

N,X(N),Y(N),IPOLR,XORIG,YORIG
FORMAT(I5,2F10.5,I5,2F10.5)

N - Number of the node.

X(N),Y(N) - The x- and y-coordinates of the node if IPOLR is blank. If IPOLR is greater than zero, a polar coordinate system is assumed. In this case, X(N) is the radial distance to the node, and Y(N) is the angle in degrees counterclockwise from the x-axis to the node; XORIG and YORIG are meaningful only for the polar coordinate option and are the x- and y-coordinates of the origin of the polar system.

NOTE 6:

The coordinates of intermediate nodes along straight element edges can and should be generated by the program. Generation is accomplished by not including such nodes in this data group. Along curved element edges, the coordinates of both intermediate nodes must be defined in such a way that the length of the curved edge is divided into thirds in close approximation. To this end, the polar coordinate option is ideal to define the nodes on circular element edges.

Group IV - Nodal Constraints

There are two parts to this data group. The first is REQUIRED, while the second is included only if there is a multiply constrained node set (NTIE not zero on card 2 of Group I data).

A. Prescribed Displacements (Required)

One card is required for each nodal constraint. This part of the data MUST BE TERMINATED BY A BLANK CARD.

```
N,NDOF,ANGLE,XDISP,YDISP
FORMAT(2I5,F10.5,2E10.5)
```

(1-5) N - Number of the constrained node.

(6-10) NDOF - 1 = the x' degree of freedom is constrained.
2 = the y' degree of freedom is constrained.
3 = both x' and y' degrees of freedom are constrained.

(11-20) ANGLE - The angle in degrees (positive counterclockwise), that the x- and y-axes must be rotated to correspond with the x'- and y'-axes. (Thus ANGLE allows x'- and y'-axes to be established for imposition of skewed constraints.)

(21-30) XDISP - Given displacement in the x'-direction, if applicable.

(31-40) YDISP - Given displacement in the y'-direction, if applicable.

NOTE 7:

Intermediate side nodes as well as corner nodes should be constrained as required by the problem. The structure must be constrained against rigid body translation or rotation. Axisymmetric problems are self-constrained in the x-direction; therefore, only rigid body motion in the y-direction must be constrained for problems of this type. If only one component of displacement is constrained at a node, the other component remains an independent degree of freedom.

When enriched elements are used to solve symmetric crack problems, a symmetry constraint should be applied to the node corresponding to the crack tip as well as to other nodes on the plane symmetry.

B. Multiply Constrained Node Data (If Needed)

There are at most three cards to define a multiply constrained node set. The first card lists the numbers of the nodes in the set in a (16I5) format. A second card (only if there are more than 16 nodes) lists the remaining nodes in the same format. The list must not exceed 30 nodes.

The last card contains:

NDOF,ANGLE
FORMAT(I5,F10.5)

(1-5) NDOF - A value of 1 causes the x' degree of freedom to be equal for the tied nodes. A value of 2 causes the y' degree of freedom to be equal for the tied nodes.

(6-15) ANGLE - As defined in Part A.

Group V - Distributed Tractions (Required)

One or two cards are required for each element edge having distributed normal and/or shear tractions. The distribution of such tractions along the element edge can be completely general; however, most problems can be treated with a constant or linear variation. Normal tractions are positive when directed into the element, and shearing tractions are positive when acting in a counterclockwise sense along the element edge. The program automatically computes the correct sign of the resulting nodal loads. Specifying surface tractions results in consistent nodal loads. This data group is TERMINATED BY A BLANK CARD which must be included even if there are no distributed tractions.

```
M1,M2,M3,M4,P1,P4,P2,P3,MORE
FORMAT (4I5,4F10.5,I5)
```

M1 through M4 - Node numbers defining the loaded element edge, given in the same counterclockwise sequence as was used to define the element; M1 is obviously a corner node as is M4.

P1 - Normal stress at Node M1.

P4 - The normal stress at Node M4; if zero or blank, P4 is set equal to P1, and a constant pressure distribution is assumed for the element edge (if P4 is zero but a constant distribution is not the case, approximate P4 with a very small number).

P2 and P3 - Normal stresses at Nodes M2 and M3; if both are blank, a linear distribution of pressure is assumed as defined by the values of P1 and P4.

MORE - If not blank, the element edge has distributed shearing stresses; these are given on an immediately following card in the form:

```
TAU1,TAU4,TAU2,TAU3
FORMAT(4F10.5)
```

The TAU's and their default settings are defined exactly as the previously described P's.

NOTE 8:

It is preferable to let the program calculate nodal loads for distributed tractions because the consistent (correct) nodal loads differ from what one would intuitively calculate. For example, for one unit of force distributed evenly over an element edge, one would expect nodal loads of 1/6, 1/3, 1/3, and 1/6. The correct consistent nodal loads, however, are 1/8, 3/8, 3/8, and 1/8. This point should be kept in mind in interpreting predicted nodal reaction forces.

NOTE 9:

The data here have been set up so that if the tractions are uniform, only P1 and/or TAU1 need be specified. If the tractions vary linearly, only P1 and P4 and/or TAU1 and TAU4 need be specified. If the loading varies more generally, all values must be specified.

NOTE 10:

Distributed tractions along the crack face for fracture applications are permissible.

Group VI - Concentrated Nodal Loads (Required)

One card for each node having a concentrated load; this data group is TERMINATED BY A BLANK CARD, which must be included even if there are no concentrated loads.

N,FX,FY
FORMAT(I5,2F10.5)

N - Number of the loaded node.

FX,FY - The x- and y-forces acting on the node, positive if acting in the positive x- and y-directions.

NOTE 11:
For axisymmetric problems, concentrated nodal loads are actually line loads which act on the complete circumference of the structure. Such loads are input on a load per radian basis. For example, if a load of 3 units per circumferential unit of length acts at a distance x from the axis of symmetry, the total force F is $(2\pi x) \times 3 = 6\pi x$. The required input load, on a per radian basis, is $F/2\pi$ or $3x$.

Group VII - Additional Data (As Required)

This data group is divided into three parts, none of which is required unless specified by nonzero values for NFRAC, IHEAT, or IPLOT on Card 2 of the Group I data.

A. Fracture Mechanics Input (If Needed)

Fracture mechanics data (required if NFRAC $\neq$ 0). These data are NOT terminated by a blank card. If NFRAC >0, complete Part 1. If NFRAC <0, complete Part 2.

1. If NFRAC has been set as a positive number, then a special singular core crack tip element is implied. In this case, only one data card is required:

XC,YC,ALPHA,RHO,NCIRC,IMODE,PO,PC
FORMAT(4F10.5,2I5,2F10.5)

XC,YC - The x- and y-coordinates of the crack tip; see Figure 2.

(21-30) ALPHA - The angle made by a line, running from within the crack toward the crack tip with respect to the x-axis. This angle is measured in degrees positive counterclockwise from the x-axis; see Figure 2.

(31-40) RHO - Radius of the crack tip core element.

(41-45) NCIRC - Number of nodes on the core element.

(46-50) IMODE - 0 = combined mode problem with full core element
1 = mode I problem with semicircular core element.

(51-70) PO,PC - If there is normal pressure along the crack face, PO is the value of the pressure at the outside of the core element, and PC is the value of the pressure at the center of the core element; between these points, the pressure is assumed to vary linearly.

2. If NFRAC has been set as -n (implying n crack tips modeled by means of enriched elements) then n cards are required, each containing:

M,IMODE,ALPHA
FORMAT(2I5,F10.5)

(1-5) M - Number of the node which corresponds to the crack tip.

(6-10) IMODE - 0 = combined mode problem, crack tip surrounded by enriched elements.

1 = mode I problem, enriched elements on only one side of crack with symmetry boundary condition imposed on Node M.

(11-20) ALPHA - As defined just previously. It is important to note that the nodal coordinates of enriched elements lying along the crack edges should be sufficiently accurate that the actual crack angle (determined from such coordinates) corresponds very closely with the angle ALPHA input here. Inaccuracy in this regard can result in computation of erroneous stress intensity factors.

B. Thermal Stress Analysis Input (If Needed)

Thermal stress analysis data are required if IHEAT ≠ 0. This part, if required, MUST BE TERMINATED BY A BLANK CARD.

Card 1 - (COEF(I),I=1,NMAT)
FORMAT(5F10.9)

The coefficients of thermal expansion for material 1 through material NMAT; NMAT is number of materials (Card 2 of Group I).

Cards 2 - NODE,TEMP(NODE) (many cards, terminated by a blank)
FORMAT(I5,F10.4)

Node number and temperature; there MUST be one such card for each corner node in the idealization. If the temperatures are not given for some (or all) pairs of intermediate side nodes, APES will automatically interpolate their values linearly from the values at the corner nodes.

In certain thermal problems, an abrupt change in temperature may exist at an interface between elements, and thermal loading is desired for elements to one side of the interface but not for those on the other. In this case, it is necessary to use two different materials, with the coefficient of thermal expansion set equal to zero for elements for which no thermal loading should be calculated.

C. Graphical Output Data (If Needed)

Graphical output data are required if IPLOT ≠ 0.

Card 1 - IDEAL,IDEFL,ICONT,IEFF,IX,IY,IZ,IS1,IS2,ITEMP,NSUBS
FORMAT(11I5)

(1-5) IDEAL ≠ 0 - Draw the structural idealization, including blowup areas if any are specified; see ICONT as follows.

(6-10) IDEFL ≠ 0 - Draw deflected structure superimposed on undeflected structure. Value is the percent of the maximum structural dimension to which the maximum deflection will be scaled on the deflected plot. Good values range from 10 to 30, depending on artistic taste.

(11-15) ICONT ≠ 0 - Draw contour plots.
If 1, draw over entire idealization only.
If 2, also over one blowup area.
If 3, also over two blowup areas.
If 4, also over three blowup areas.

(16-20) IEFF ≠ 0 - Draw contours of effective (von Mises) stresses.

(21-25) IX ≠ 0 - Draw contours of x-stresses.

(26-30) IY ≠ 0 - Draw contours of y-stresses.

(31-35) IZ ≠ 0 - Draw contours of z-(out-of-plane) stresses.

(36-40) IS1 ≠ 0 - Draw contours of maximum (in absolute value) principal stresses.

(41-45) IS2 ≠ 0 - Draw contours of minimum (in absolute value) principal stresses.

(46-50) ITEMP ≠ 0 - Draw contours of temperatures.

(51-55) NSUBS - Number of parts into which the element edges will be broken to produce subelements for plotting contours (default 6, maximum 11). The larger this number, the smoother the contours, at the expense of additional computing time.

If ICONT = 0 or 1, no further data are required.

If ICONT >1, then (ICONT-1) additional cards are required. Each card can carry as many as 20 element numbers which specify a group of elements selected for blowup plotting. These numbers are given in a (20I4) format. If there are less than 20 elements in a blowup area, leave the excess entries blank. Note that this is the only place in the input data in which integer numbers are input in other than an I5 type of format.

OUTPUT

Modifications to APES can be readily made to provide any kind of output normally associated with the finite element method. To keep the volume of output within reason, however, it is presently limited to the following:

1. An echo of the input data as converted for use by the computer, i.e., input values as well as generated values.

2. When applicable, the nodal loads generated from input distributed tractions.

3. Where applicable, stress intensity factors and displacements of the crack tip(s). For the core crack tip element, the displacements are given parallel and perpendicular to the line of the crack, i.e., in local coordinates. For enriched-element crack tip models, the crack tip displacements are given in global coordinates along with displacements of all other nodes in the idealization.

4. Global displacements of all nodes.

5. Reaction forces at constrained nodes.

6. Global (ε_x, ε_y, ε_z, γ_{xy}) strains at each node of each element, if not suppressed.

7. Global stresses at each node of each element, if not suppressed.

8. Global stresses averaged at the nodes.

9. Principal stresses and angles and the von Mises effective stress at the nodes, based on the global stresses averaged at the nodes. Angles are measured in degrees counterclockwise from the x-axis. The von Mises stress is calculated as

$$\sigma_{vm} = (\sigma_x^2+\sigma_y^2+\sigma_z^2-\sigma_x\sigma_y-\sigma_y\sigma_z-\sigma_z\sigma_x+3\sigma_{xy}^2)^{1/2}$$

where σ_z is the out-of-plane stress for plane strain problems and the circumferential stress for axisymmetric problems; it is zero for plane stress problems.

10. The node having the highest von Mises stress; stresses and strains are not calculated within a singular core element nor are they highly accurate at enriched element nodes which are near the crack tip.

Graphical output available has already been described in the preceding section. Examples are provided in the appendix.

ERRORS AND TIPS

Do not attempt to use the APES program without first studying and understanding both this manual and Reference 2. Many errors are made by beginners simply because they do not carefully read and understand the input instructions. First, the beginner should solve a small, simple problem, having a known solution before applying the program to more practical situations. In this way, familiarization with the input data and confidence in the accuracy of the program will be quickly obtained.

Following is a list of the more common errors that anyone can inadvertently make:

1. Failing to justify entries on the right in I or E formats.

2. Failing to include a blank card when required to terminate a series of data cards.

3. Incorrectly listing the nodes that define the elements; failing to list the nodes in a counterclockwise sense; failing to begin the list with the crack tip node if the element is an enriched crack tip element.

4. Using incorrect nodal coordinates, causing elements to lose quadrilateral shape or to be turned "inside out." This is particularly likely to happen when the geometry of an idealization is changed to provide a modified problem without correctly updating all affected nodal coordinates.

5. Repeating a node number at two different places in an idealization or omitting a node number. Numbering only corner nodes where possible can reduce the probability of this type of error.

6. Failing to prevent rigid body motion of the element assemblage because of inadequate boundary conditions.

7. Failing to impose correct boundary conditions. This is extremely dangerous because the user thinks one problem is being solved while APES accepts the boundary conditions as "legal" and proceeds to solve a completely different problem.

8. Incorrectly applying distributed or point loading, which is dangerous for the same reason given in Item 7.

A reasonable attempt has been made in APES to provide error diagnostics, particularly in the graphical preprocessor. It is not possible, however, to detect logically many of the errors that can be made. For this reason, both input and output should be carefully checked to ensure that the correct problem is actually solved.

The worst thing that can occur is the diagnostic phrase "SINGULAR MATRIX . . ., etc.," after the input data has passed through the data checking preprocessor with no indication of error. Do not make the mistake of thinking that something is wrong with the program if the data appear to be correct; they just look that way. In this event, there is a high probability that the error will be difficult to find. Verify that rigid body translation and rotation are precluded by the boundary conditions and that the material numbers of the elements do not exceed the total number of input materials. If the problem is still unresolved, check with someone more experienced in the use of the program. It is fortunate that problems of this sort cease to occur once some experience in using the APES program has been gained.

The choice of a good finite element mesh for a given problem is an art in which only experience can produce a high degree of competence. The examples given in References 2 and 3 provide some guidance but, clearly, they cannot be extrapolated to the limitless number of practical problems which may arise. Fortunately, the high-order, 12-node element is "forgiving" because of its inherent high accuracy; thus, results having acceptable engineering accuracy will probably be obtained, even if the element mesh is considerably less than ideal. Only a few general suggestions can be made with regard to selecting an element mesh.

1. Concentrate elements in areas of anticipated high gradients in stress. In other areas, keep in mind that a rectangular (undistorted) 12-node element is capable of modeling exactly a stress field containing up to quadratic terms in the polynomial which describes its spatial distribution; in other words, few elements will generally be needed.

2. Avoid elements with interior corner angles that are either very large or very small. Extreme distortion, in some cases, can lead to results which are seriously in error[6*]; see the table. As a rule of thumb, keep corner angles in the range from 45 to 135 degrees, the closer to 90 degrees the better. When using curved element edges, attempt to minimize the included angle. For example, at least two elements should be used to span a 90-degree curved radius.

3. Although very good accuracy has been observed in thin shell applications employing single elements (with high aspect ratios) through the shell thickness, there is no known guarantee that this success will carry into other applications. Therefore, elements with very high aspect ratio should be used only with caution.**

4. If curved element edges are being used to model curved boundaries, be sure that the intermediate nodal coordinates are given so that the curved edge is divided into thirds to close approximation. Misplaced intermediate nodes can lead to highly inaccurate results and can even result in a strain and stress singularity as described by Pu and Hussain.[10] The polar coordinate option available for defining nodal coordinates is ideal for defining such curved edges. For straight element edges, let APES generate the coordinates of intermediate nodes.

---

*Documentation of extreme cases of loss of accuracy due to highly skewing elements or to employing highly curved edges has been reported informally in DTNSRDC Structures Department Technical Note m-2, "Accuracy Loss in Distorted Isoparametric Elements," by D.A. Hopkins and L.N. Gifford (Sep 1978).

**A study of accuracy as a function of element aspect ratio and shape has been reported informally in DTNSRDC Structures Department Technical Note, "A Further Study of Accuracy Loss in Distorted Isoparametric Elements," by J.B. Sickles and L.N. Gifford (in press). It was concluded that under the worst of conditions, accuracy is a function of mesh density only (and not element shape) if element skew angles are held in the range of 45 to 135 degrees and if element aspect ratios are held at 8 or less.

5. If distributed tractions are applied to the structure, do not attempt to calculate the equivalent nodal loads. These are not as intuition would suggest. Let the program calculate these loads, and rest assured that they will be correct.

6. When using enriched 12-node elements to model cracked structures, increased accuracy in the stress intensity factors is not achieved by reducing the enriched element sizes toward zero.[11] In fact, enriched elements may be surprisingly large; either Reference 2 or 3 should be consulted for examples. It appears that there is an "ideal" size (not extremely large or extremely small) for a given fracture problem; however, the size that is "just right" cannot be precisely defined. Fortunately, excellent accuracy is obtained over a broad range of enriched element sizes.

DESCRIPTIONS OF ASSOCIATED PROGRAMS

As of the end of 1978, there are four different programs associated with the APES system, all of which accept input data as described in the preceding. These are:

1. CHECK - The graphical (CALCOMP) preprocessor, which performs logical checks of the input data and plots the idealization with element numbers, node numbers, etc. Present dimensioning is for 200 elements, which represents a very large problem for the high-order element used in the APES system. This program loads and executes in approximately 100 000 octal (33 000 decimal) words of core on Control Data Corporation Series 6000 computers.

2. APES - A small version of the analysis, limited to 50 or fewer elements and 400 or fewer nodes. This size is adequate for the majority of engineering problems which have been encountered thus far. The program loads and executes in approximately 105 000 octal (35 000 decimal) words of core on Control Data 6000-series computers.

3. KINGKONG - A larger version of the analysis, limited to 100 or fewer elements and 700 or fewer nodes. The program loads and executes in 130 000 octal (45 000 decimal) words of core on Control Data 6000-series computers.

4. KINGKONG II - An even larger version of the analysis, limited to 200 or fewer elements and 1400 or fewer nodes. Unlike APES and KINGKONG which permit only 40 nodes on the wave front when solving the master stiffness equations (sometimes leading to difficulty in numbering elements to meet this limitation), KINGKONG II permits as many as 80 nodes on the wavefront. It loads and executes in 240 000 octal (82 000 decimal) words of core on Control Data 6000-series computers.

The graphical postprocessor is contained as an overlay in the analysis programs APES, KINGKONG, and KINGKONG II.

Execution time for the analysis program (with no plotting of results requested) runs approximately 2 to 3 seconds per element on the Control Data 6400 computer. Execution on more advanced Control Data Corporation machines is considerably faster. If plotting of computed results is desired, additional computation time will be expended. In fact, if a great deal of plotting is requested, the plotting may take more computer time than the analysis.

All programs are written in FORTRAN and compile on the Control Data FTN (FORTRAN Extended) compiler. All programs are "overlayed" and, because of the available 60-bit word size (14 significant figures), are in single precision. The analysis programs are about 5000 lines long, and about 800 of these are "comments." There are 59 programs, subroutines, or functions. The plotting overlays of the programs utilize CALCOMP-provided subroutines, and all are standard. Because the programs are written expressly for Control Data Corporation equipment, conversion to other computing systems and plotters, while fairly straightforward, is probably not a 1-day task.

ACKNOWLEDGMENTS

Professor Peter D. Hilton of Lehigh University has been directly involved in developing and improving the APES program since its infancy. His significant contributions (many of which are described here) are gratefully acknowledged. The idealization for the graphical output examples was provided by Ms. Mary Donovan of the Center.

APPENDIX
EXAMPLES OF GRAPHICAL OUTPUT

To illustrate some of the graphical features provided with the APES system, the axisymmetric example of a fairly thick sphere penetrated at its appex by a short, thick-walled, cylindrical tube is considered.

Figure 5 shows the region to be idealized. This view of the right-hand cross section has been created by drawing element edges unconnected to other elements, thus giving a good first visual check of the correctness of the input data. Only the upper half of the sphere has been considered. The centerline of the structure runs vertically just left of the cross section shown. The loading is uniform, external, hydrostatic pressure.

Figures 6 through 8 show the actual breakdown of the structure into 41 elements and numbering of the elements and nodes. (Plots of this type may be made as large as desired to increase clarity.) Two layers of elements have been used through the spherical section when one would be practically as good. In fact, the entire problem is probably overdone in number of elements; however, this is better than having too few. As a consequence, the predicted stresses are anticipated to be extremely accurate.

The critical region of the structure is around the intersection of sphere and cylinder. This area was chosen for zoomup stress contour plotting as shown in Figure 9.

Figure 10 shows the magnified inward deflection of the structure (solid line) superimposed over the undeflected structure (dashed line) during external loading. In this plot, 15 percent of the maximum structural dimension (IDEFL=15 in the graphical data input, Group VII, Part C) was chosen for scaling the peak deformation.

Contour plots of von Mises effective stresses across both the entire structure and the zoomup area are shown in Figures 11 and 12. Similar contour plots for circumferential (z) stresses are shown in Figures 13 and 14; for maximum principal stresses, in Figures 15 and 16.

It is worth mentioning that the example problem was set up and solved by a newly recruited college graduate, who had no previous experience in applying the finite element method. Computation cost, including a data checking run and plots of the results, was less than $30. Total engineering cost was considerably higher because of the work required to define the faired, curved surfaces. As a rough measure, an experienced analyst would require less than 2 days to complete the problem from perception to output. It is interesting to speculate how a senior structural analyst may have approached this problem, the time that may have been consumed, the accuracy that would have been achieved, and the manner in which the results would have been presented as little as 15 years ago when the finite element method was almost unheard of.

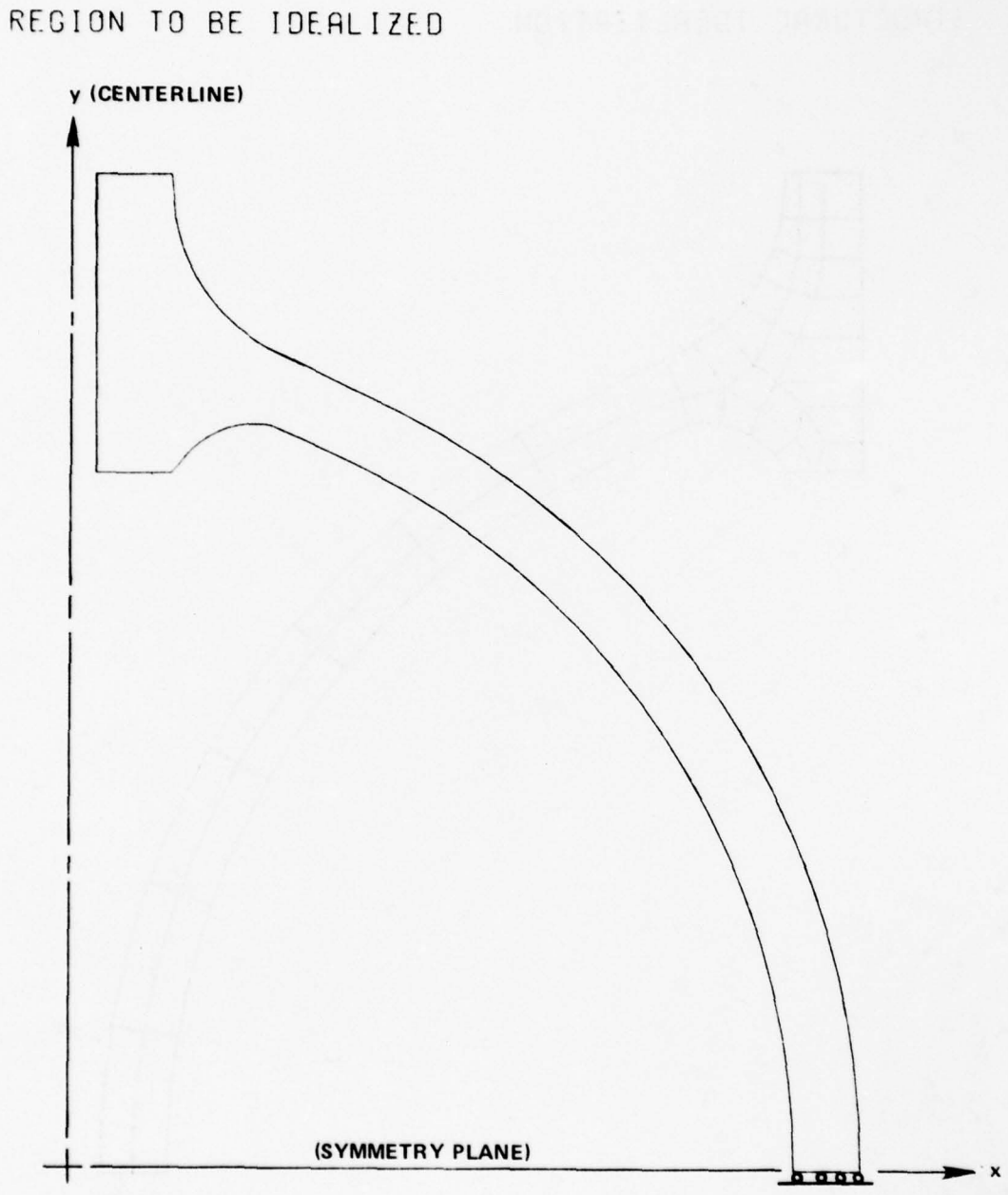
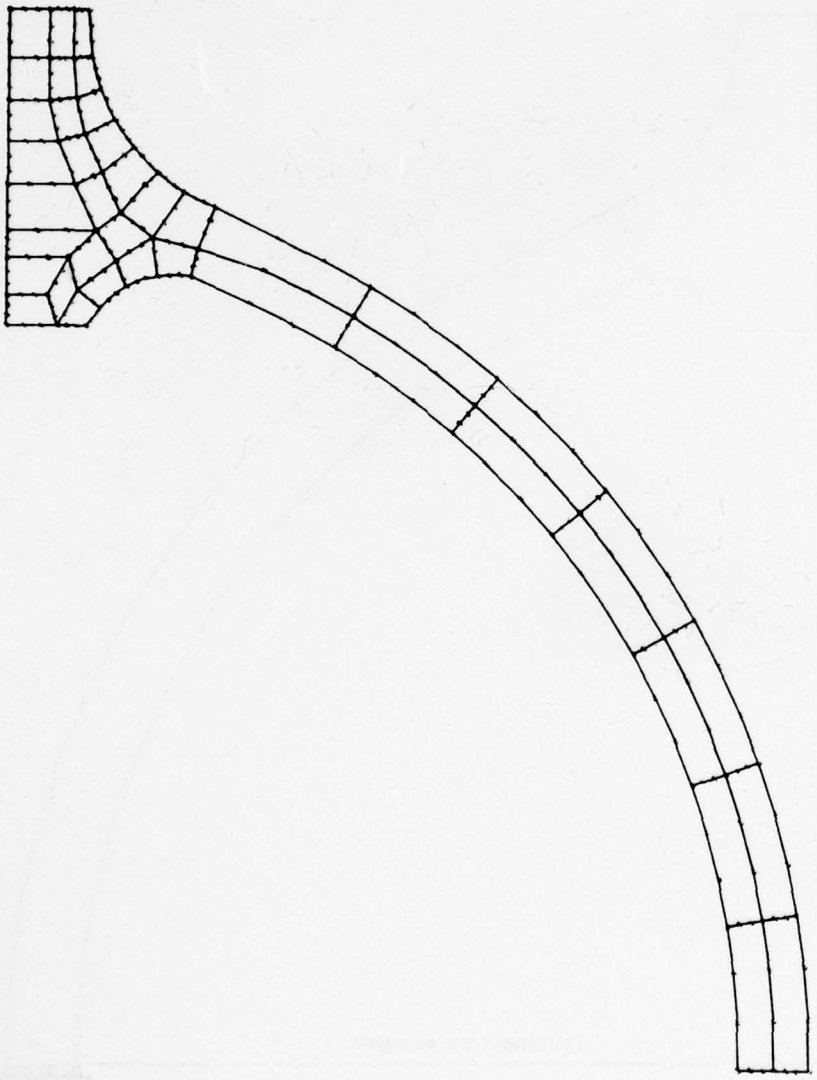


Figure 5 - Outline of Symmetric Half of Example Problem

STRUCTURAL IDEALIZATION



CYLINDER-SPHERE INTERSECTION

Figure 6 - Element Mesh for Example Problem

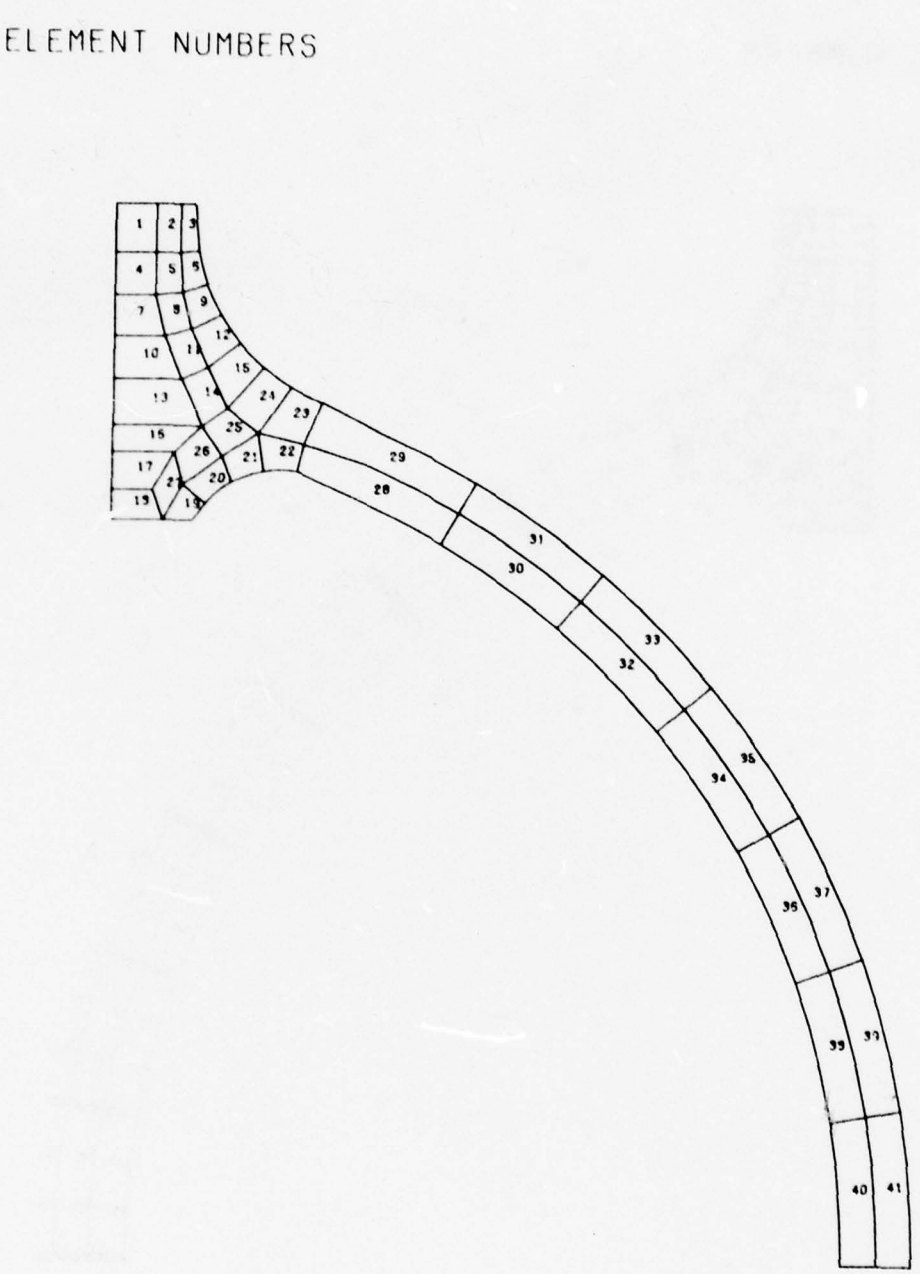
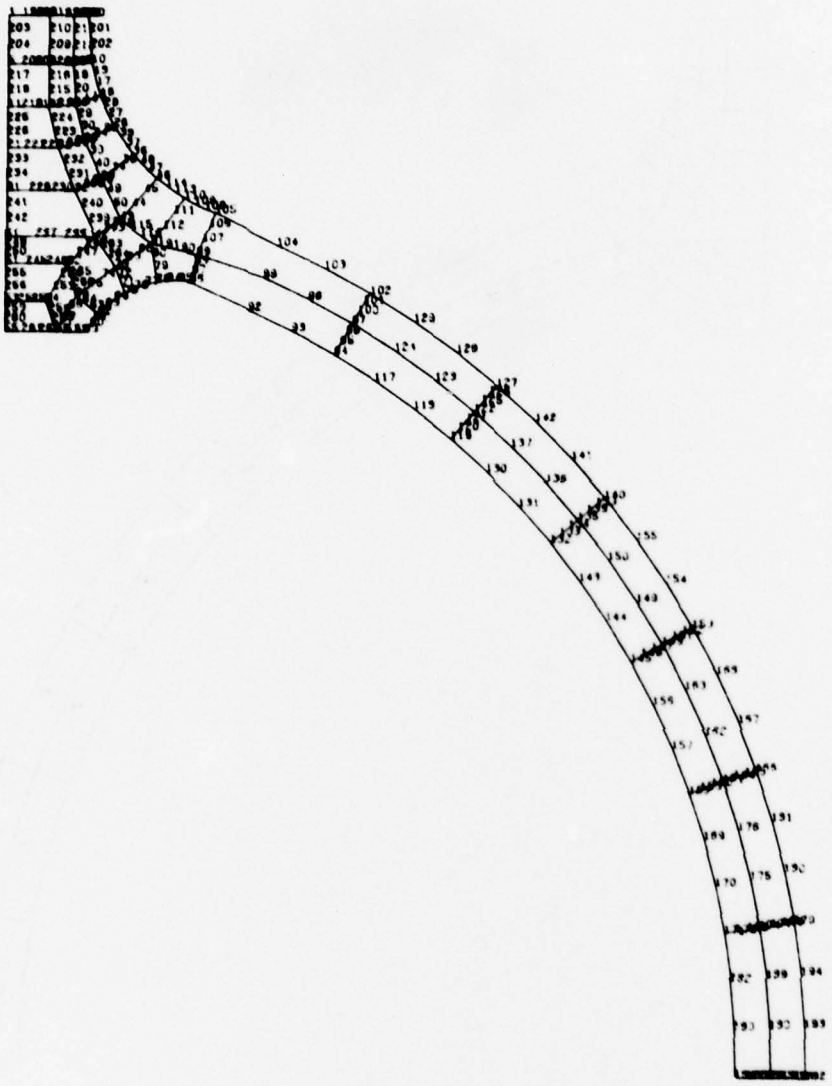


Figure 7 - Element Numbering for Example Problem

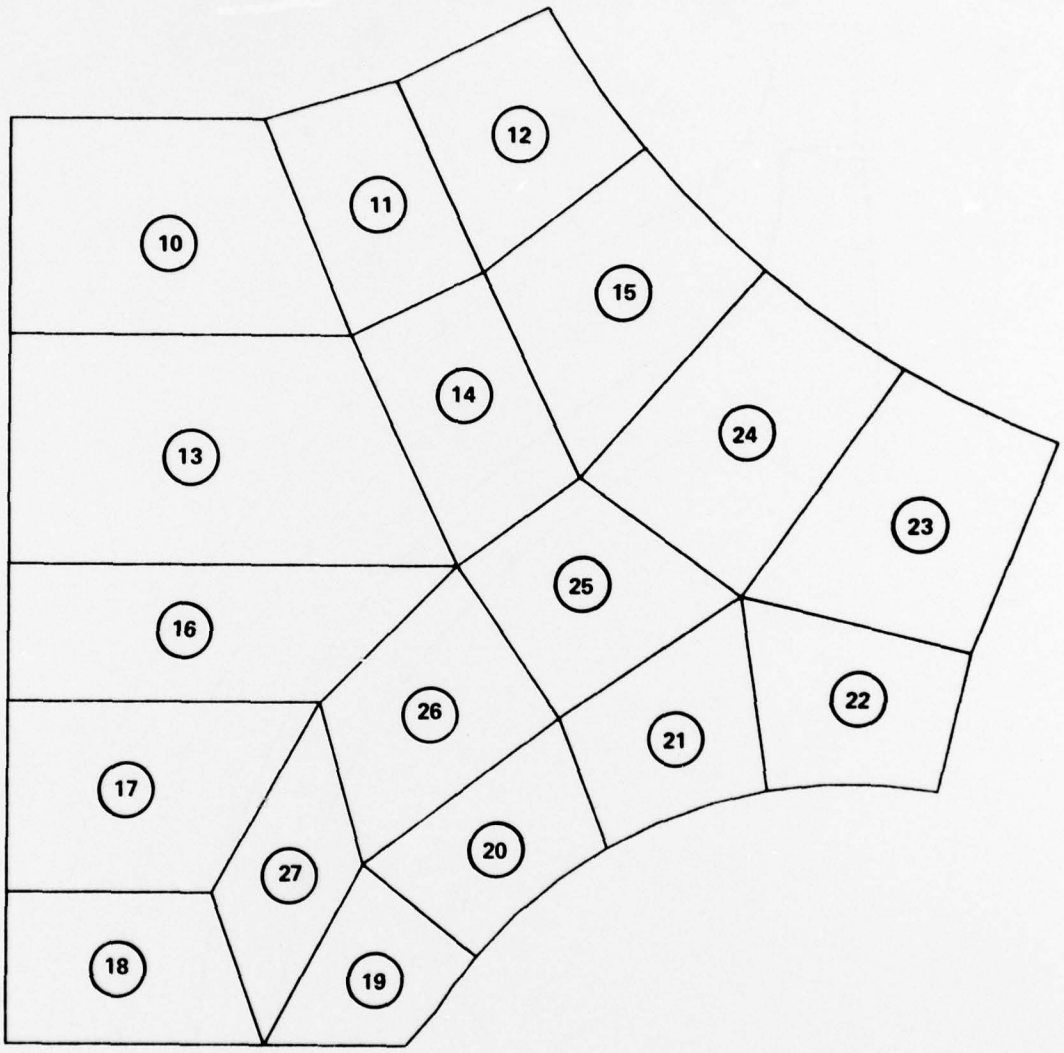
NODE NUMBERS



CYLINDER-SPHERE INTERSECTION

Figure 8 - Node Numbering for Example Problem

IDEALIZATION IN ZOOM-UP AREA



CYLINDER-SPHERE INTERSECTION

Figure 9 - Stress Critical Region of Example Problem

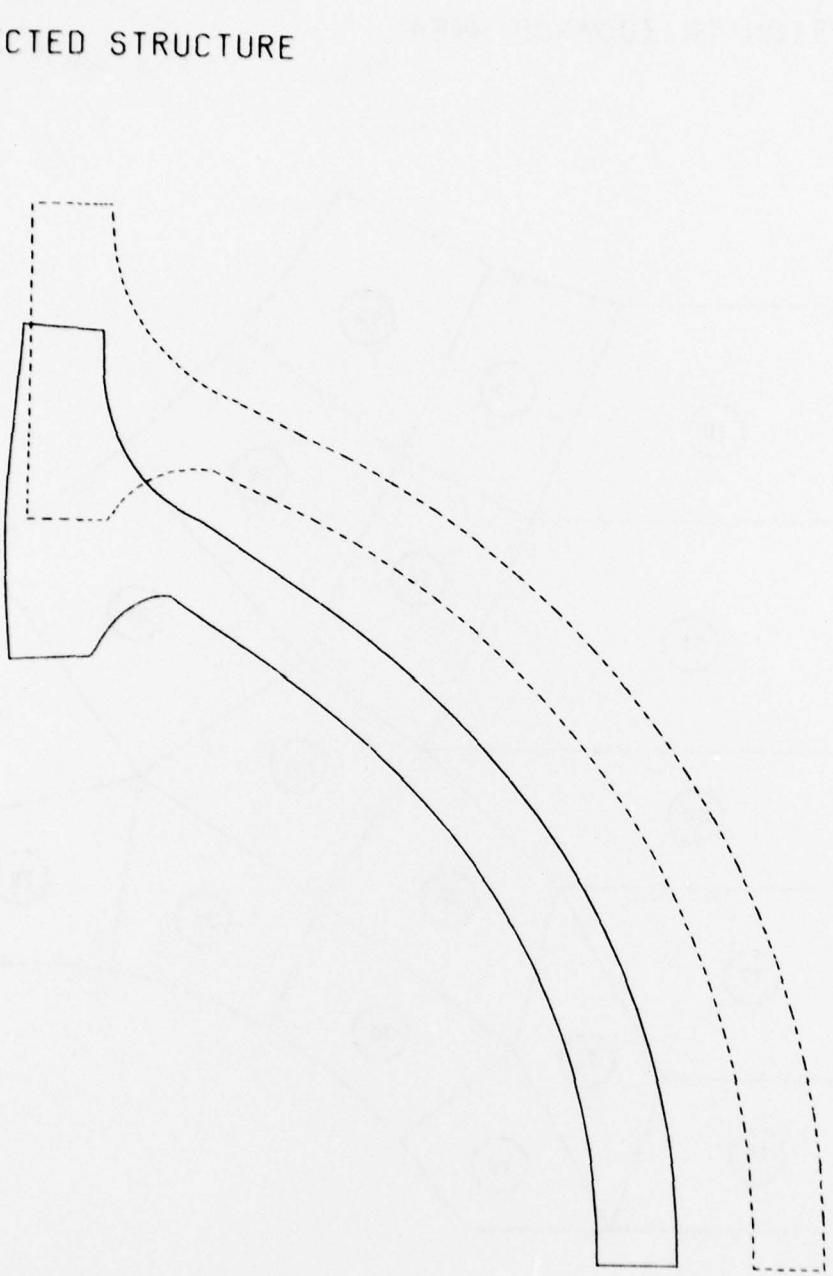


Figure 10 - Deformation Pattern of Example Problem

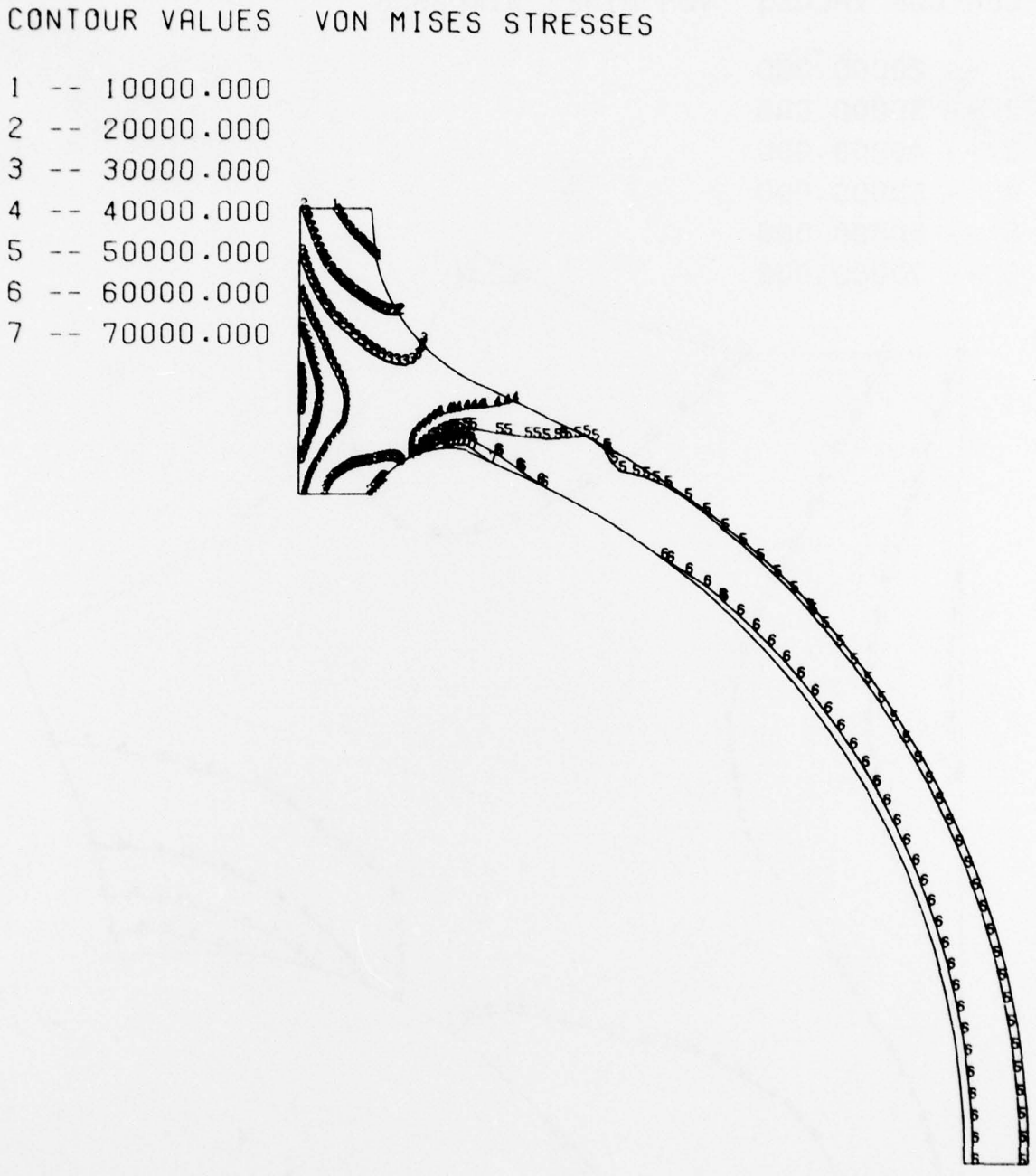


Figure 11 - von Mises Stress Contours over Example Problem

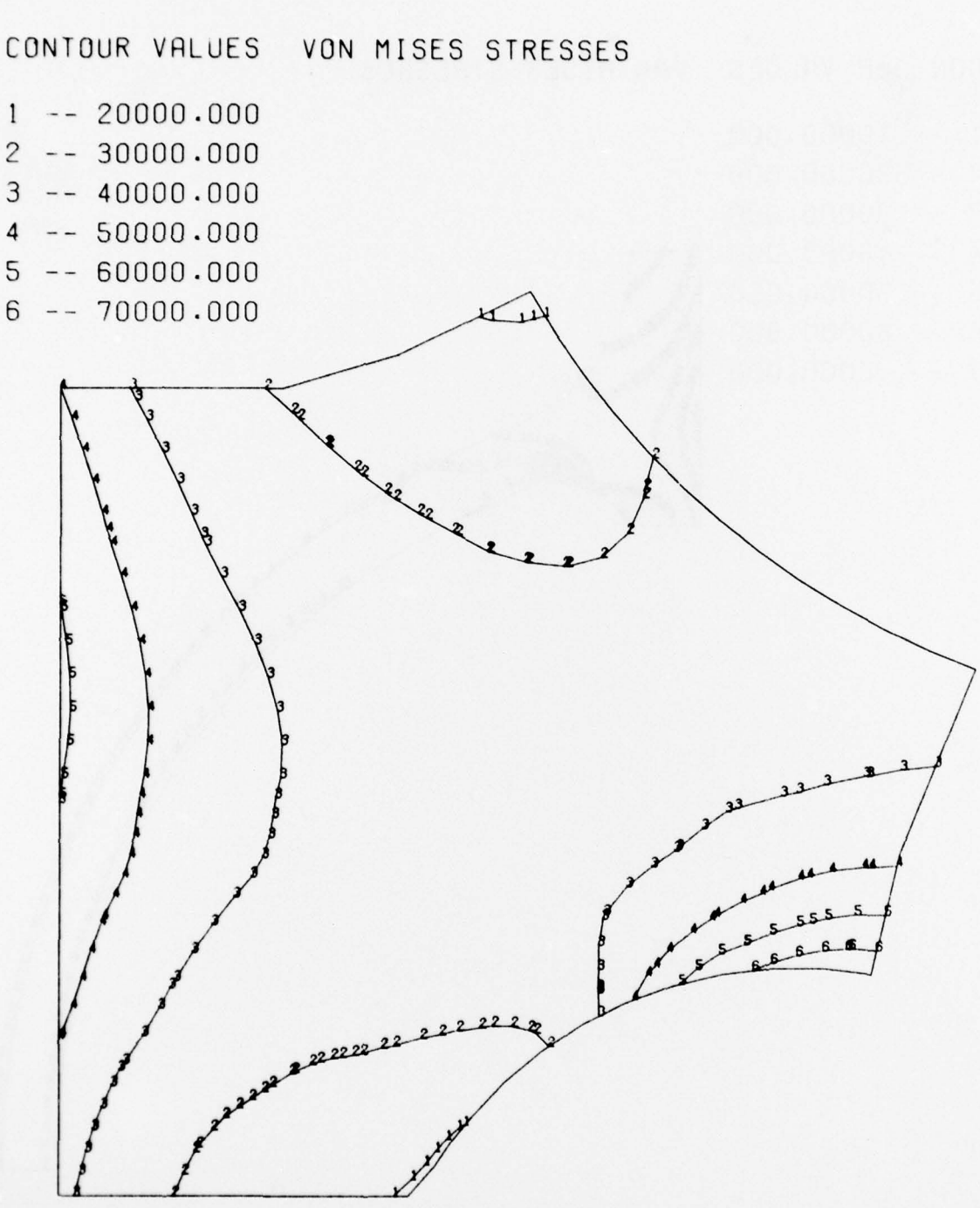


Figure 12 - von Mises Stress Contours over Critical Region

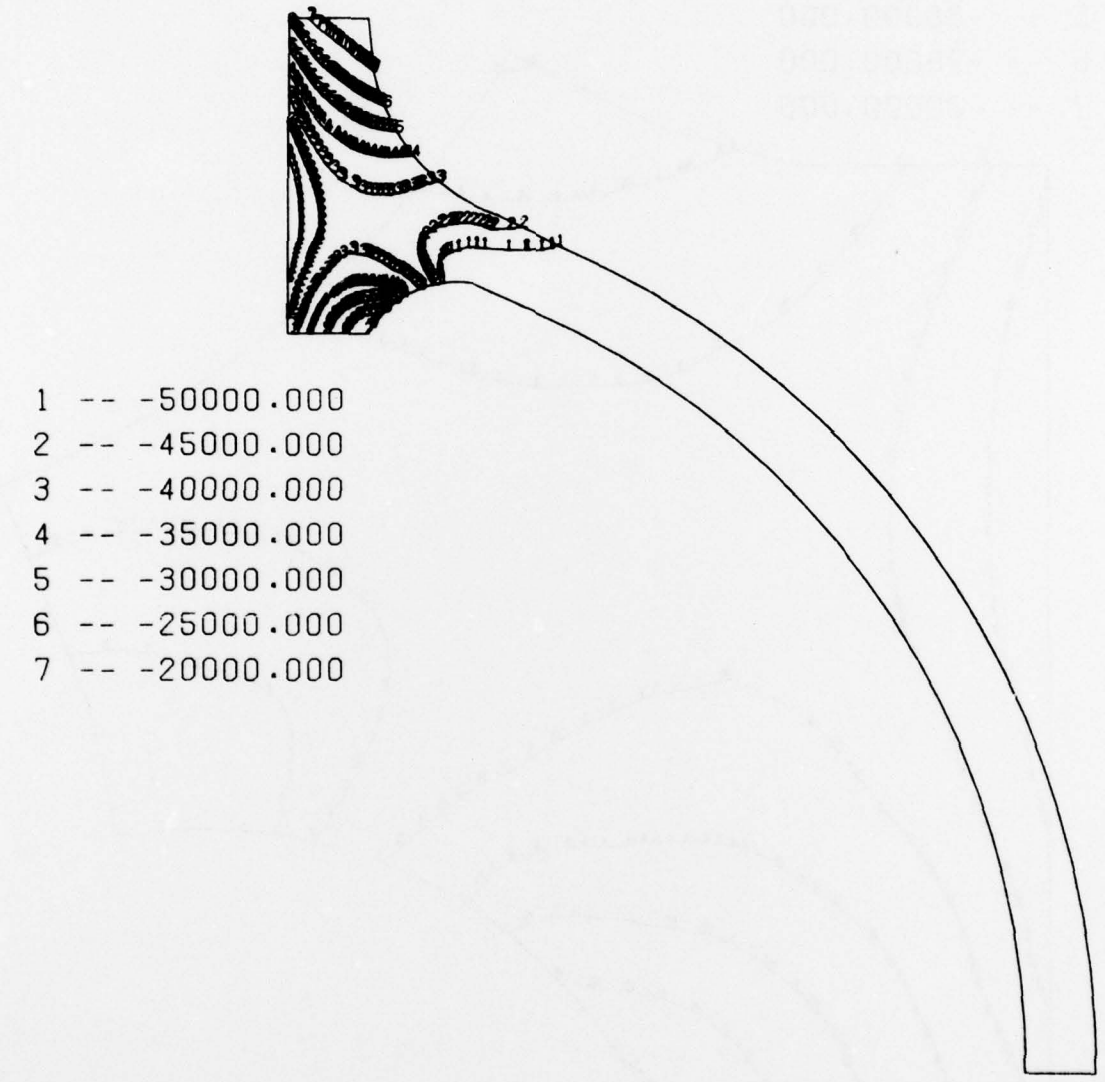


Figure 13 - Circumferential Stress Contours over Example Problem

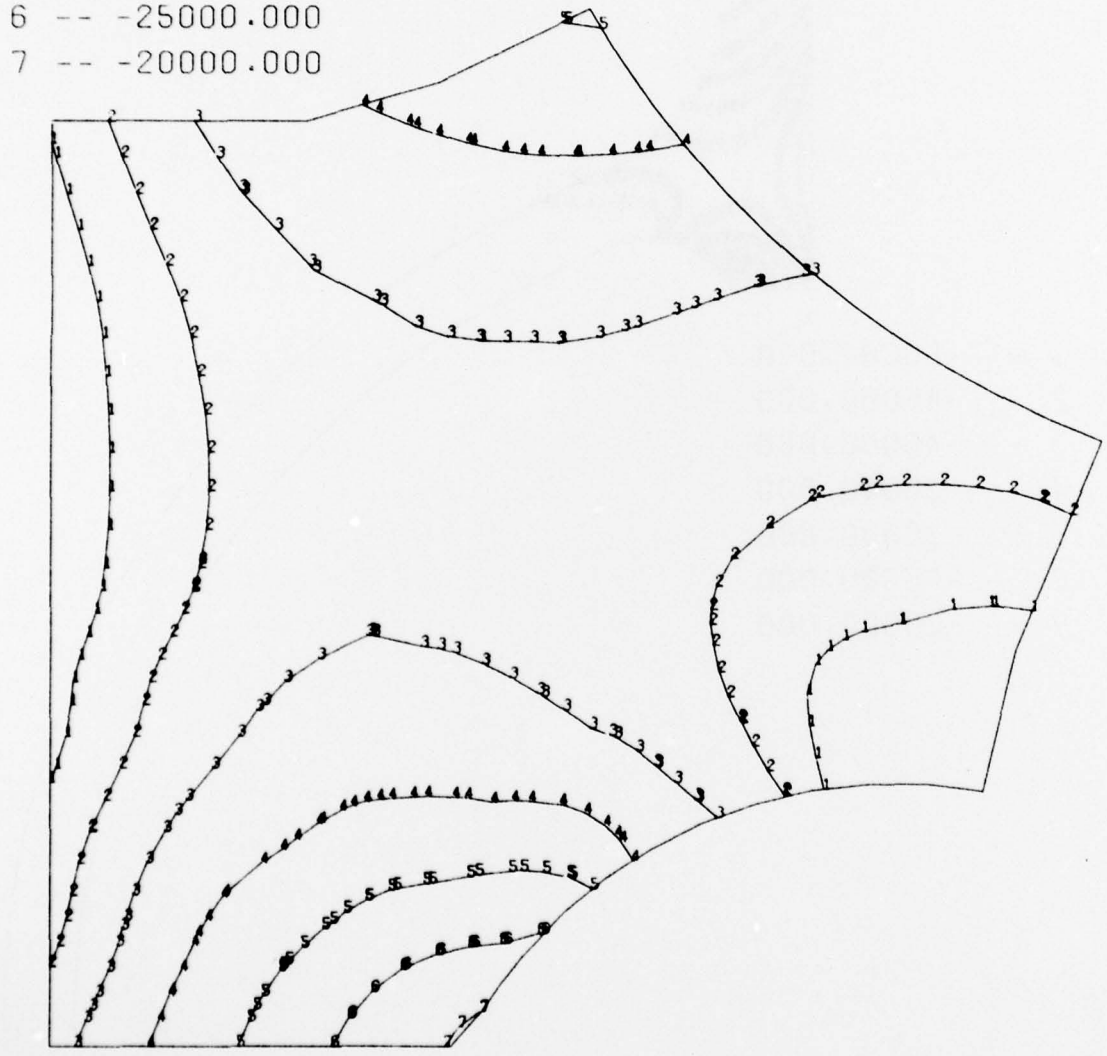


Figure 14 - Circumferential Stress Contours over Critical Region

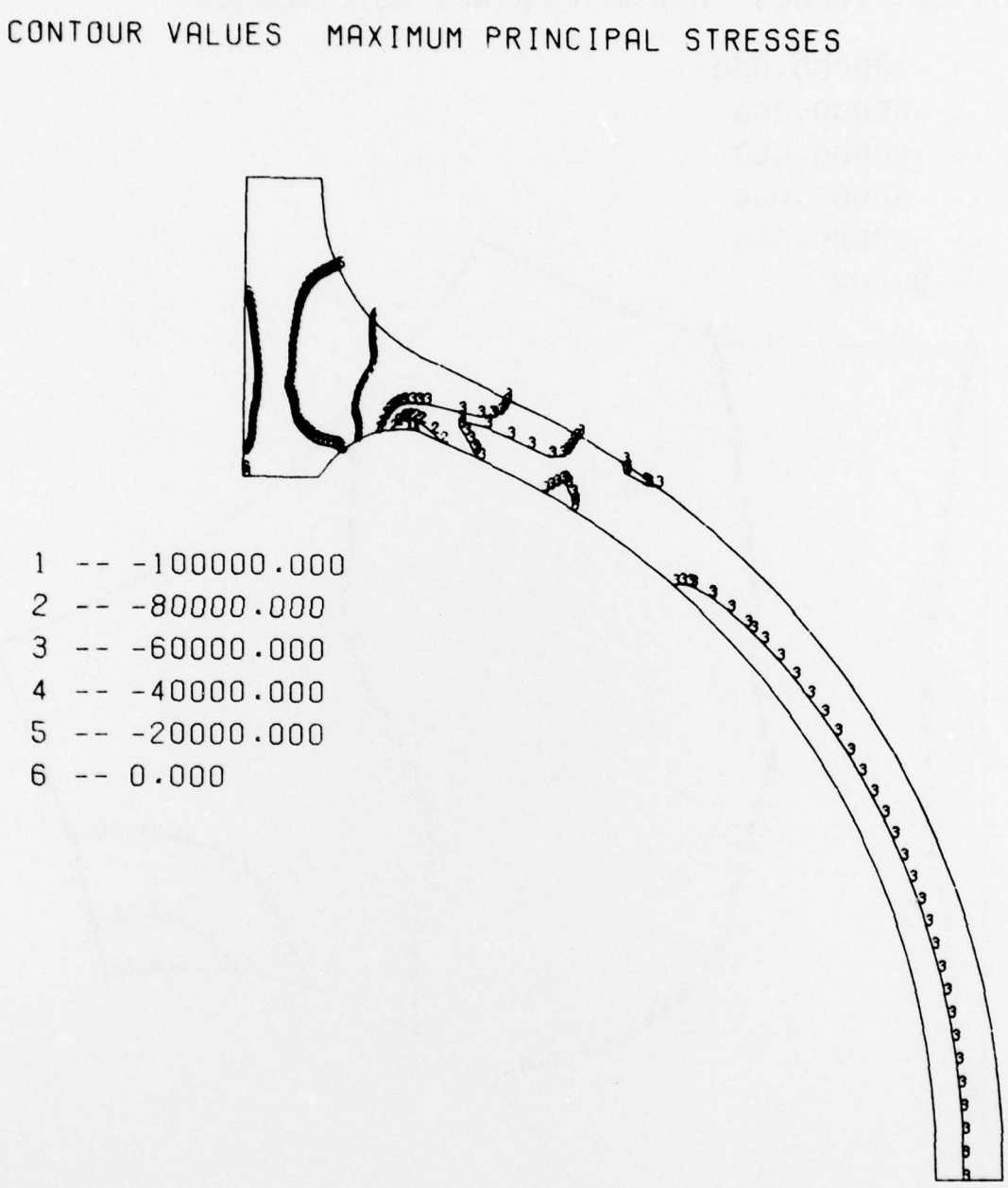


Figure 15 - Principal Stress Contours over Example Problem

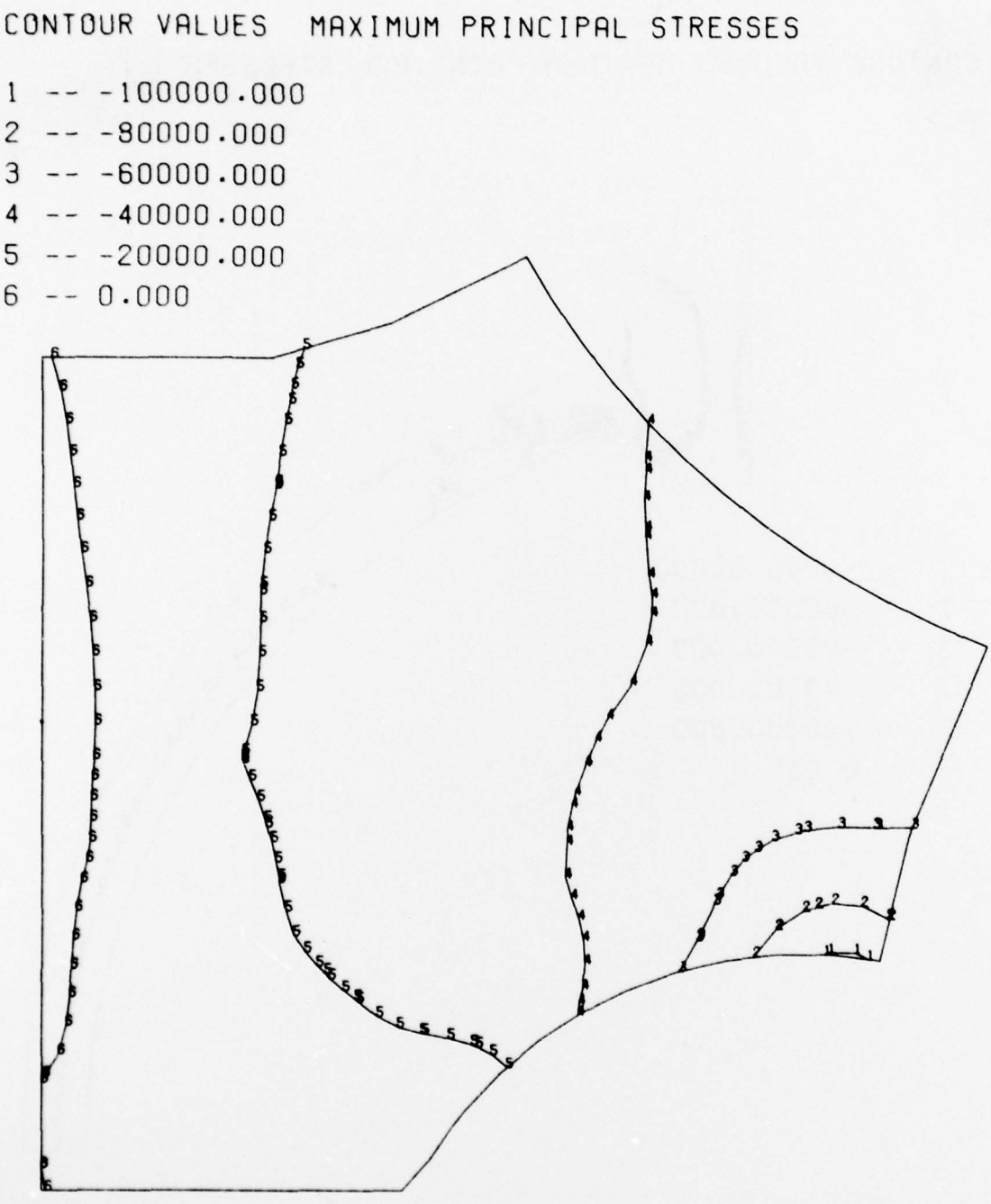


Figure 16 - Principal Stress Contours over Critical Region

REFERENCES

IJNME is International Journal for Numerical Methods in Engineering.
DTNSRDC is David W. Taylor Naval Ship Research and Development Center.

1. Zienkiewicz, O.C., "The Finite Element Method in Engineering Science," McGraw-Hill Book Company, London (1971).

2. Gifford, L.N., "APES-Second Generation Two-Dimensional Fracture Mechanics and Stress Analysis by Finite Elements," DTNSRDC Report 4799 (Dec 1975).

3. Gifford, L.N. and P.D. Hilton, "Stress Intensity Factors by Enriched Finite Elements," Engineering Fracture Mechanics, Vol. 10, No. 3, pp. 485-496 (1978).

4. Hilton, P.D. et al., "Finite Element Fracture Mechanics Analysis of Two-Dimensional and Axisymmetric Elastic and Elastic-Plastic Cracked Structures," DTNSRDC Report 4493 (Nov 1974).

5. Hinton, E. et al., "Local Least-Squares Stress Smoothing for Parabolic Isoparametric Elements," IJNME, Vol. 9, pp. 235-256 (1975).

6. Gifford, L.N., "More on Distorted Isoparametric Elements," IJNME (in press).

7. Bäcklund, J., "On Isoparametric Elements," IJNME, Vol. 12, pp. 731-732 (1978).

8. Stricklin, J.A. et al., "On Isoparametric versus Linear Strain Triangular Elements," IJNME, Vol. 11, pp. 1041-1043 (1977).

9. Irons, B.M., "A Frontal Solution Program for Finite Element Analysis," IJNME, Vol. 2 (1970).

10. Pu, S.L. and M.A. Hussain, "The Collapsed Cubic Isoparametric Element as a Singular Element for Crack Problems," IJNME (in press).

11. Kacprzynski, J.J., "Application of Some Finite Element Programs to Linear Elastic Fracture Mechanics," National Research Council of Canada Laboratory Technical Report LTR-ST-956 (Nov 1977).

INITIAL DISTRIBUTION

Copies	
1	DDR&E, Tech Lib
1	CNO, OP987 R&D Plans Div
4	CHONR
	1 ONR 102
	1 ONR 430
	1 ONR 465
	1 ONR 474
3	NAVMAT
	1 MAT 08E
	1 MAT 08T1
	1 MAT 08T23
8	NRL
	1 Code 6300
	1 Code 6360
	1 Code 6380
	1 Code 6382
	1 Code 6384
	1 Code 8406
	1 Code 8430
	1 Code 8433
4	NAVSEA
	1 SEA 035
	1 SEA 03511
	1 SEA 03521
	1 SEA 0353
1	USNA
1	NAVPGSCOL
1	USNROTC & NAVADMIN, MIT
1	NADC
1	NOSC
1	NWC
1	NCSC
1	CEL

Copies	
1	NSWC, White Oak
1	NSWC, Dahlgren
1	NUSC
1	NAVAIR, 320
1	NAVFAC, 03
1	NAVSHIPYD PTSMH
1	NAVSHIPYD CHASN
1	NAVSHIPYD MARE
1	SUPSHIP GROTON
1	SUPSHIP NEWPORT NEWS
1	SUPSHIP PASCAGOULA
12	DDC
1	ASD/ENFSF, WPAFB
1	AFDL, WPAFB
1	ASIAC (AFFDL/FBR)
2	WATERVLIET ARSENAL
	1 Library
	1 Dr. S.L. Pu
2	AMMRC
	1 Library
	1 Dr. T.P. Rich
3	NASA/Langley
	1 Library
	1 Dr. J.C. Newman
	1 Dr. R.E. Fulton
3	NASA/Lewis
	1 Library
	1 Dr. J.E. Srawley
	1 Dr. B. Gross

Copies	
1	NASA/Goddard
1	NASA Scientific Tech Info Ofc
2	NUC Reg Comm 1 Library 1 Dr. J.R.N. Rajan
1	USCG
1	MARAD
2	BUSTAND, Wash 1 Library 1 Dr. J.T. Fong
2	BUSTAND, Boulder 1 Library 1 Dr. D.T. Read
1	University of Arizona Dr. R.H. Gallagher
3	Battelle Memorial Institute 1 Library 1 Dr. G.T. Hahn 1 Dr. E.F. Rybicki
1	Brown University Dr. J.R. Rice
1	Boston University Dr. I. Fried
2	Carnegie Melon Inst 1 Dr. J.L. Swedlow 1 T.A. Cruse
1	Colorado State Univ Dr. F.W. Smith
1	Univ of Connecticut Prof A.J. McEvily
2	Franklin Inst 1 Library 1 Z. Zudans

Copies	
2	George Washington Univ 1 J.D. Lee 1 A.K. Noor
1	Georgia Inst of Tech Dr. J.M. Anderson
1	Harvard Univ Dr. J.W. Hutchinson
2	Johns Hopkins Appl Phys Lab 1 Library 1 G. Daley
2	Lehigh Univ 1 Dr. G.C. Sih 1 Dr. R.P. Wei
1	Univ of Maryland Dr. G.R. Irwin
1	Univ of Massachussetts Dr. W.A. Nash
1	MIT Dr. K. Masabuchi
1	Michigan Tech Univ Dr. V.W. Snyder
1	Stanford Research Institute
1	Southwest Research Institute J.R. Maison
1	Univ of Texas Dr. R.S. Dunham
1	VPI&SU Dr. G.W. Swift
3	Univ of Washington 1 APL 1 Dr. A.S. Kobayashi 1 Dr. A.F. Emery
1	WHOI

Copies	
1	Yale Univ Dr. D.M. Parks
1	Air Products & Chemicals M.G. Zellner
1	ALCOA (Pitts) J.G. Kaufman
2	Battelle Pacific Northwest Labs 1 M.C.C. Bampton 1 G.H. Beeman
1	Boeing Aerospace Company (Seattle) 1 A.V. Viswanathan
2	Brookhaven National Lab 1 Dr. M. Reich 1 D. Gardner
1	General Dynamics Corp Electric Boat Div
2	General Electric Co (Schenectady) 1 Dr. C.F. Shih 1 H. Saunders
1	Ingalls Shipbldg Corp
1	Lockheed Palo Alto Res Lab
1	NNSB&DD Co
1	Northrop Corp (Hawthorne) 1 J.R. Yamane
4	Sandia Laboratories (Albuquerque) 3 Dr. P.D. Hilton 1 Dr. S.E. Benzley
1	U.S. Steel Corp (Monroeville) J.M. Barsom
1	Welding Research Council
3	Westinghouse Elec Corp (Pittsburgh) 1 Dr. W.K. Wilson 1 Dr. D.E. McCabe 1 Dr. N.E. Dowling
1	Weston Components & Controls W.R. Hartman

CENTER DISTRIBUTION

Copies	Code
1	11
1	17
1	1702
2	1706S
1	1707
2	172
5	1720.1
1	1720.2
3	1720.3
60	1720.4
1	1720.5
2	1720.6
1	173
1	1730.5
1	1730.6
1	174
1	1740.5
1	177
1	1805
1	1844
1	28

Copies	Code
1	2814
1	282
10	5211.1 Reports Distribution
1	522.1
1	522.2